Heat Transfer and Flow Characteristics Study in a Low Emission Annular Combustor

Teddy Sedalor

Thesis submitted to the Faculty of
Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of
Master of Science
in
Mechanical Engineering

Dr. Srinath V. Ekkad
Dr. Danesh K. Tafti
Dr. Bob West

December 7, 2009
Blacksburg, Virginia

Keywords: Combustor liner cooling, Swirler, Infrared Thermal Imaging, Dry Low Emissions

Copyright 2009, Teddy Sedalor
ABSTRACT

Heat Transfer and Flow Characteristics Study in a Low Emission Annular Combustor

Teddy Sedalor

Modern Dry Low Emissions (DLE) combustors are characterized by highly swirling and expanding flows that makes the convective heat load on the combustor liner gas side difficult to predict and estimate. A coupled experimental-numerical study of swirling flow and its effects on combustor liner heat transfer inside a DLE annular combustor model is presented. A simulated scaled up annular combustor shell was designed with a generic fuel nozzle provided by Solar Turbines to create the swirl in the flow. The experiment was simulated with a cold flow and heated walls. An infrared camera was used to obtain the temperature distribution along the liner wall. Experimentally measured pressure distributions were compared with the heat transfer results. The experiment was conducted at various Reynolds Numbers to investigate the effect on the heat transfer peak locations and pressure distributions. A CFD study was performed using Fluent and turbulence models and used to corroborate and verify the experimental results. Results show that the heat transfer enhancement in the annulus has slightly different characteristics for the concave and convex walls. Results also show a much slower drop in heat transfer coefficient enhancement with increasing Reynolds number compared to can combustors from a previous study. An introductory study of the effect of a soft wall on the heat transfer on the combustor liner is also presented.
ACKNOWLEDGEMENTS

This thesis is one of the many goals I set for myself; I hereby take this opportunity to thank anyone who helped me realize this goal. Without their various forms of support and guidance this work would not be complete. First and foremost, I would like to thank my advisor Dr Srinath Ekkad for giving me the opportunity to work as his graduate student. He provided guidance and constructive criticism during my research to make it a success. I would also like to thank Solar Turbines Inc for providing the funding for the project.

The suggestions of my committee members; Dr Tafti and Dr West were greatly appreciated. Dr Tafti and Sunil presented the CFD validation of the work presented in this thesis. I am also indebted to Santosh Abraham for the immense guidance he provided in the early stages of my research. I am grateful for my lab mates Chris Le blanc, Justin Lamont and Pritish Parida for lending a hand physical and mentally.

My immediate family consisting of my wife Jessica, and my little man Cyril, supported and encouraged me outside the lab to keep me going, and I deeply appreciate their help, and love them dearly. Finally, I am very grateful to my parents Monica and George Sedalor for getting me to the United States to further my education, their efforts and encouragement has gone a long way.
TABLE OF CONTENTS

CHAPTER 1: INTRODUCTION ................................................................................................ 1
  1.1 Combustor ............................................................................................................................. 3
  1.2 Pollutant Emissions ............................................................................................................... 9
  1.3 Solar Dry Low Emissions (DLE) Combustor ................................................................. 13
  1.4 Swirler ................................................................................................................................. 16
  1.5 Focused Cooling/Backside Cooling .................................................................................... 17
  1.6 Soft Wall ............................................................................................................................. 19
  1.7 Literature Survey ................................................................................................................. 20
  1.8 Experimental Objectives ..................................................................................................... 22

CHAPTER 2: EXPERIMENTAL SETUP ............................................................................... 24
  2.1 Inlet Air Supply/Blower ...................................................................................................... 26
  2.2 Swirler ................................................................................................................................. 28
  2.3 Combustion Chamber .......................................................................................................... 29
  2.4 Surface Wall Heater (constant heat flux) ............................................................................ 30
  2.5 Infrared Thermal Imaging System .................................................................................... 330
  2.6 Personal Daq USB Data Acquisition Module ................................................................. 34
  2.7 NetScanner 9816/98RK Rackmount Ethernet Intelligent Pressure Scanner ....................... 35

CHAPTER 3: EXPERIMENTAL METHODOLOGY ........................................................... 37
  3.1 Uncertainty Analysis ............................................................................................................. 42
LIST OF FIGURES

Figure 1.1 Schematic of gas turbine engine ................................................................. 2

Figure 1.2: Drawing of an annular combustion chamber ................................................. 4

Figure 1.3: Schematic of a combustor with detailed components ..................................... 5

Figure 1.4: Different combustor liner cooling techniques ............................................... 7

Figure 1.5: Typical Combustor Liner Wall Film Cooling ............................................... 8

Figure 1.6: Influence of Primary-Zone temperature on CO and NOx emissions ............... 11

Figure 1.7: Solar Turbines MARS Low-NOx Combustor ................................................. 14

Figure 1.8: Comparison of a conventional combustor to SoloNox .................................. 15

Figure 1.9: Focused Cooling ............................................................................................ 18

Figure 2.1: Views of Test Setup and Swirler .................................................................. 25

Figure 2.2: Cincinnati Fans blower and VTAC 9 controller ............................................. 26

Figure 2.3: Graph of motor frequency against convex combustor pressure ................. 27

Figure 2.4: Graph of motor frequency versus concave combustor pressure ..................... 27

Figure 2.5: Schematic of the generic swirler produced by solar .................................... 28

Figure 2.6: Test setup showing combustion chamber/test chamber ............................... 29

Figure 2.7: Schematic of Surface Heater ....................................................................... 30

Figure 2.7a: Heat loss as a function of temperature ....................................................... 31

Figure 2.8: FLIR SC640 Infrared Camera ...................................................................... 33

Figure 2.9: Sample combustor image taken by IR Camera ........................................... 33
Figure 2.10: OMB-DAQ-54 Personal Daq ................................................................. 35

Figure 2.11 NetScanner multi-channel pressure acquisition system ................................. 36

Figure 3.1 Test section configuration with three swirlers .................................................. 39

Figure 3.2: Schematic of concave and convex test sections ............................................. 41

Figure 3.3: Combustor front panel with softwall .............................................................. 41

Figure 4.1 Sample IR image of liner ................................................................................. 46

Figure 4.2 Processed image of IR picture ....................................................................... 46

Figure 4.3 2-D wall temperature and heat transfer coefficient (W/m\(^2\)-K) distributions on the concave liner wall at Re=420000 ................................................................. 47

Figure 4.4 2-D wall temperature and heat transfer coefficient (W/m\(^2\)-K) distributions on the convex liner wall at Re=420000 ................................................................. 48

Figure 4.5 Comparing convex and concave surface heat transfer enhancement along the combustor liner ............................................................................................... 49

Figure 4.6 Effect of Reynolds number on Nusselt number augmentation for concave surface ... 50

Figure 4.7 Effect of Reynolds number on Nusselt number augmentation for convex surface ..... 51

Figure 4.8 Nusselt number augmentation on concave liner wall ...................................... 52

Figure 4.9 Nusselt number augmentation on convex liner wall ........................................ 53

Figure 4.10 2-D axial velocity profile in the annular combustor ....................................... 54

Figure 4.11 Transverse section of axial flow profile ......................................................... 55

Figure 4.12 Contours of streamlines in the computational domain .................................... 57
Figure 4.13 Averaged axial velocity along liner................................................................. 58

Figure 4.14 Contours of normalized axial velocity in meridional plane in computational domain ......................................................................................................................................................................................................................................................... 59

Figure 4.15 Superposition of axial velocity and nusselt augmentation .......................... 60

Figure 4.16 Contours of normalized turbulent kinetic energy in meridional plane in computational domain ......................................................................................................................................................................................................................................................... 61

Figure 4.17 Variation of peak heat transfer augmentation with Reynolds number .......... 62

Figure 4.18 Effect of softwall on the local peak heat transfer ........................................... 64

Figure 4.19 Comparison of heat transfer coefficient profile on liner with and without softwall ......................................................................................................................................................................................................................................................... 65
NOMENCLATURE

$D_h$ hydraulic diameter of annular comustor

$D_{sw}$ mean diameter of swirler

$G_m$ axial flux of angular momentum

$G_t$ axial trust

$A$ area of resistance heater

$R$ electrical resistance

$Q$ heat generation

$I$ electrical current

$V$ electrical voltage

$h$ local convection heat transfer coefficient

$k$ thermal conductivity

$\nu$ kinematic viscosity

$Nu$ Nusselt number

$Re$ Reynolds number

$S_N$ swirl number

$T$ air temperature
CHAPTER 1: INTRODUCTION

During the past three decades, environmental regulations governing pollutant emissions have become increasingly stringent with the requirement of lower levels of nitrous oxide (NOx), carbon monoxide (CO), and unburned hydrocarbons (UHC). At the same time, increases in turbine inlet temperatures are being pushed to attain higher overall efficiency in power generation turbines. Both the desire for better efficiency and the need for lower emissions have reduced the amount of cooling air that the combustion engineer has available for combustor liner cooling. As combustors are designed to reduce emissions, there is insufficient liner cooling available since more air is utilized in the premixing process and reaction zones to maintain a low temperature profile. Due to this requirement, the effectiveness of backside cooling techniques involving impingement, convection, or surface enhancement techniques becomes more critical. The lack of knowledge of the local gas side heat transfer distribution on the combustor liner makes effective cooling of liners more difficult. The results from this study will help in understanding and predicting swirling flow effects on the local convective heat load to the combustor liner and thus support the development of more effective cooling schemes to maintain/improve combustor durability.

The gas turbine is one of the most efficient means of producing power either through direct shaft power, or thrust produced from the high momentum exhaust gasses. Hot combustion gases are used to drive the turbine in a gas turbine engine. The first few gas turbine engines were aimed only at performance, without the consideration of emissions.

Fig 1.1 shows a basic layout of a gas turbine engine. The fundamental thermodynamic principles that make gas turbine engines feasible are based on the properties of the working fluid.
The working fluid, air, is compressed and then heated with the addition and burning of fuel. The energy that is released from the air expanding through the turbine is more than the energy that is required to compress the air. This energy acquired is amplified by compressing the fluid to a higher pressure and increasing the temperature at the turbine inlet. The temperature will therefore reach over or near the failure point of the turbine materials, necessitating extensive cooling of the parts exposed to the hot flow. The coolant air is extracted from the compressor section before it reaches the combustion chamber. It is sent to the parts of the engine that are in the hot gas path, i.e. mainly combustor liner and the turbine blades, which need to be cooled using innovative designs.

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. To obtain higher efficiency,
the firing temperatures in gas turbine combustors are raised. However, to achieve the stringent emissions levels in terms of CO and NOx production, it is required to maintain the combustion zone temperatures as low as possible. The desire for higher efficiency and the requirement for lower emissions have reduced the amount of cooling air that the combustion engineer has available for the combustor liner. Modern DLE combustors are characterized by highly swirling and expanding flows that make estimation of the convective heat load on the gas side very difficult.

The study focuses on the interaction between the hot swirling gases and the liner wall within a gas turbine combustor. Better understanding of the heat transfer process from the gases to the combustor liner is essential so that effective usage of the available coolant air may be planned. Therefore with detailed analysis of the local heat transfer rates, a more effective cooling system on the backside of the combustor liner can be designed

1.1 Combustor

The combustor is designed to burn a mixture of fuel and air and to deliver the resulting gases to the turbine. The gas temperature must not exceed the allowable structural temperature of the turbine. About one half of the total volume of air entering the burner mixes with the fuel and burns. The rest of the air-secondary air- is simply heated or may be thought of as cooling the products of combustion and cooling the burner surfaces. Combustion chambers may be of can, annular or can-annular type. The one focused on for this study is the annular combustor. Figure 1.2 shows a typical annular combustor.
Figure 1.2: Drawing of an annular combustion chamber [1]

The turbine engine main burner system consists of three principal elements: the inlet diffuser, the dome and snout, and the liner. This study is focused on the combustor liner. The combustion process is contained by the liner. The liner also allows introduction of intermediate and dilution airflow and liner’s cooling airflow. The liner must be designed to support forces resulting from pressure drop and must have thermal resistance capable of continuous and cyclic high temperature operation [1]. Figure 1.3 shows the schematic of a typical combustor liner.
Effective control of the air distribution in, around and through the main burner is vital to the attainment of complete combustion, stable operation, correct burner exit temperature profile, and acceptable liner temperatures for long life. Primary air is the combustion air introduced through the dome of the burner and through the first row of liner air holes. This air mixes with the incoming fuel, producing the locally near-stoichiometric mixture necessary for optimum stabilization and operation. To complete the reaction process and consume the high levels of primary zone CO, H-, and unburned fuel, intermediate air is introduced through a second row of liner holes. The reduced temperature and excess oxygen cause CO and H concentrations to decrease. In contemporary systems, the dilution air is introduced at the rear of the burner to reduce the high temperature of the combustion gases. The dilution air is used to carefully tailor
exit temperature radial and circumferential profiles to ensure acceptable turbine durability and performance [1].

Present day technology and techniques are capable of pushing the thermal efficiency higher than ever and increasing the engine power output with better control of air-fuel ratio (AFR) and flame temperatures. With increase in the thermal efficiency and power output, turbine inlet temperature will also increase consequently from 1700K to 2200K, which is higher than the blade material melting temperature at around 1850K. To overcome this, various solutions have been tried and tested; using thermal barrier ceramic coating (TBC) and 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 durability of components. However, excess removal of heat from gas turbines is unfavorable as it will reduce the turbine inlet temperature which in turn reduces the thermal efficiency of the gas turbine. Therefore a proper design of cooling system is important to compensate for both thermal efficiency and destructive heat loads.

Cooling air must be used to protect the burner liner and dome from the high radiative and convective heat loads produced within the burner. This air is normally introduced through the liner such that a protective blanket or film of air is formed between the combustion gases and the liner. The effectiveness of the cooling technique is quantified by the cooling effectiveness φ, defined by

\[
\phi = \frac{T_g - T_m}{T_g - T_c}
\]

(1.1)

Where Tg, Tm, and Tc are the mainstream gas, average metal, and cooling air temperatures, respectively. Some techniques used to cool combustor liners are shown in Figure 1.4 below.
Film cooling technique has been used as the most popular cooling system for the past few decades. Figure 1.5 shows the typical film cooling system used in conventional turbine combustor. Here, the secondary coolant air is injected through holes or slots into the mainstream air flow in the primary combustion chamber to form a thin separate coolant air film or jacket which prevents the mainstream hot air from direct contact with the combustor liner walls. This leads to lower flame temperature and higher liner pressure drop which promotes better mixing.
Therefore with film cooling we can attain better fuel atomization and eliminate localized hot spots.

Figure 1.5: Typical Combustor Liner Wall Film Cooling [2]

The main drawback with film cooling is that the secondary coolant air stream will alter the air fuel ratio by mixing itself with the mainstream air flow. It also causes a non-uniform radial temperature distribution and premature failure of materials due to thermal stresses. 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 gets relatively low such that all chemical reactions cannot be possible resulting in unburned hydrocarbons. At the same time the temperature of air at the center core of the primary zone will get so high that it results in higher NOx emissions. These hot regions can generate 25-30 ppm NOx and the relatively cooler regions near the liner results
in higher CO in the exhaust. Thus, even when the overall combustion zone is maintained below the NOx formation threshold temperature range (1500-1650°C), the NOx levels will still be affected due to the center core region (as shown in Figure 1.5). Therefore we have an undesirable situation with the worst overall combined emissions levels of both CO and NOx. This has led to a lot of research intended to improve the temperature uniformity within the combustion primary zone.

1.2 Pollutant Emissions

Gas turbine combustion is a steady flow process in which a hydrocarbon fuel is burned with a large amount of excess air, to keep the turbine inlet temperature at an appropriate value. This was generally a clean process with the exception of the need to eliminate smoke from the exhaust. Recently, however, control of emissions has become probably the most important factor in the design of gas turbines, as causes and effects of industrial pollutants become better understood and the population of gas turbines increases.

The basic principles of combustion can be described on a thermodynamic basis as shown in Equation 1.1 below.

\[
C_{xH_y} + nO_2 \rightarrow aCO_2 + bH_2O
\]  

(1.2)

Where

\[a = x, b = \left(\frac{y}{2}\right) \text{ and } n = x + \left(\frac{y}{4}\right)\]
Each kilogram of oxygen will be accompanied by 3.29kg of nitrogen, which is normally considered to be inert and appear unchanged in the exhaust; at the temperature in the primary zone, however, small amounts of oxides of nitrogen are formed. The combustion equation assumes complete combustion of the carbon to CO$_2$, but incomplete combustion can result in small amounts of carbon monoxide (CO) and unburned hydrocarbons (UHC) being present in the exhaust. Thus the exhaust of any gas turbine primarily consists of CO$_2$, H$_2$O, O$_2$ and N$_2$. The pollutants appearing in the exhaust will include oxides of nitrogen (NOx), carbon monoxide (CO) and unburned hydrocarbons; any sulfur in the fuel will results in oxides of sulphur (SOx). All these pollutant have an adverse effect on the environment, ranging from smog and acid rain caused by NOx to the fatal effect of inhaling CO. This has led to stringent restrictions on emissions from all types of power plants [1].

The single most important factor affecting the formation of NOx is the flame temperature; this is theoretically a maximum at stoichiometric conditions and will fall off at both rich and lean mixture. NOx emissions can be more than halved by reducing the flame temperature from 1900K to 1800K. Unfortunately, while NOx could be reduced by operating well away from stoichiometric, this results in increased formation of both CO and UHC, as shown in Figure 1.6.

Inlet air temperature, residence time in the primary zone, equivalence ratio, and combustion pressure might also affect NOx emissions production level. However, primary-zone flame temperature is still regarded as the most significant contributor to NOx emissions issues. The rate of formation of NOx varies exponentially with flame temperature, so the key to reducing NOx is reducing the flame temperature. The residence time for the formation of NOx should also be minimized. The primary zone flame temperature for most conventional
combustors ranges from 1000K at low power condition to 2500K at high power conditions. Figure 1.6 shows that when the combustion temperature is below 1670K, CO and UHC are formed appreciably due to incomplete combustion from the lack of adequate oxygen and also due to the dissociation of $2CO$; whereas at temperature higher than 1900K, excessive amounts of NOx are produced. This essentially leaves us with a narrow band of temperature range between 1670K and 1900K where the CO and NOx emissions levels are within the tolerable values.

![Figure 1.6: Influence of Primary-Zone temperature on CO and NOx emissions](image)

Figure 1.6: Influence of Primary-Zone temperature on CO and NOx emissions [3]
Even though the overall averaged flame temperature is properly controlled within the desirable temperature range 1670K to 1900K, there are going to exist localized hot and cold spots that can result in the local temperatures in separate regions within combustion zone to rise higher than 1900K, escalating NOx emissions, and to drop lower than 1670K increasing CO and UHC respectively. This defeats the purpose of narrow band flame temperature control and seriously affects the pollutant emissions. 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. Due to this requirement, the effectiveness of backside cooling techniques involving impingement, convection, or surface enhancement techniques becomes more critical. Due to longer operating cycles for power turbines, combustor liner needs to meet durability targets of 30000 hours. To avoid liner failure from over-temperature, it is extremely important to accurately quantify the liner heat load in the lean premixed combustor environment. This means that cooling techniques for the low NOx combustor liner requires more backside cooling and less or almost no film cooling [3].

There are various viable ways to reduce pollutant emissions in gas turbines. One method is using water or steam injection to decrease flame temperature. This gives the inlet airflow a higher mass flow which increases power. The downside of this method is that, demineralized water which is the water used to reduce corrosion is scarce and expensive in some regions, also the thermal efficiency is drastically reduced and even though NOx emissions are reduced, CO and UHC emissions increase. Another method is catalytic reduction, in this method a catalyst is used together with small amounts of ammonia resulting in the conversion of NOx to N₂ and H₂O. The catalytic reaction only occurs in a limited temperature range (400F-600F), so this method is
used only with waste heat recovery applications which results in extremely low emissions. Disadvantages are high capital costs and the storage of noxious fluids. Finally dry low NOx systems, which are system without water injection, are still being researched and developed to reduce emissions. The combustors are designed with various techniques like fuel staging, multiple burners, inclusion of swirler and using lean and premix combustion systems. This system must be cooled well and in a way not to interfere with the emissions. This is the solution being pursued in this paper.

1.3 Solar Dry Low Emissions (DLE) Combustor

Dry Low Emissions Combustors are at the forefront of combustor design due to their ability to reduce NOx, CO and UHC emissions to as low as 9ppm. Two of the major performance criteria in the design of dry low-NOx combustors for stationary gas turbines are 1) Meeting the emissions requirements and controlling the variation of emissions levels across the load range of the engine. 2) To achieve stable combustion at all operating conditions, superior system response to rapid load changes and acceptable levels of combustion.

Solar Turbines is among the pioneers in the development of dry low-emissions combustors for industrial gas turbines [2].

The DLE combustors utilize 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. DLE combustor does not use water or steam injection into the main chamber. In order to efficiently maintain the combustion flame temperature at a favorable 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, DLE combustors implement a complex array of proprietary fuel-air-nozzles and combustion system geometry. Figure 1.7 shows an example of modern dry low NOx emission combustor used in Solar MARS engines designed by Solar Turbines.

![Figure 1.7: Solar Turbines MARS Low-NOx Combustor](image)

The new DLE combustor regulates the combustor airflow and pilot fuel flow over the entire engine-operating map. The need to control these parameters has further complicated the control system design compared to that of a conventional gas turbine. Below 50% load the combustor airflow is managed similar to that of a conventional engine. Above 50% rated load, Solar's DLE engine enters the "low emissions mode," altering either the bleed valves or inlet
guide vanes to keep the combustion primary zone temperature within a particular range. Solar's DLE gas turbine controls utilize the turbine inlet temperature as a reference of the primary zone temperature in order to control the bleed valve or inlet guide vane position as a function of turbine load [2]. Figure 1.8 below compares a conventional combustor to a SoloNOx combustor by Solar Turbines Inc, as can be seen, the flame temperature is significantly reduced but still yields the same turbine inlet temperature, the fuel is premixed to reduce local hot spots, and the shape of the combustor reduces the residence time which also reduces the NOx emissions. Based on the general requirement and challenges of combustors and in order to compensate for pollutant emissions standards and better efficiency, the modern dry low emissions combustors are believed to be the best solution for the next generation combustors.

![Image of conventional and SoloNOx combustors comparison](image)

**Figure 1.8: Comparison of a conventional combustor to SoloNOx**
1.4 Swirler

The primary zone airflow pattern is an important factor in flame stability. One of the common features among all the different types of airflow patterns employed is the creation of a toroidal flow reversal that entrains and recirculates a fraction of the hot combustion products to mix with the incoming air and fuel. These 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. An effective way of inducing flow recirculation in the primary zone is by putting in a swirler in the dome around the fuel injector. Vortex breakdown arising by the swirling flow causes recirculation in the core region when the amount of rotation imparted to the flow is high. The swirl components produce strong shear regions, high turbulence and rapid mixing rates. These characteristics of swirling flows have been used to control the stability and intensity of combustion and the size and shape of the flame region [2].

Air swirlers are widely used in both tubular and annular combustors. The two main types of swirlers are axial and radial. They are often fitted as single swirlers, but are also sometimes mounted concentrically providing either co-rotating or counter-rotating airflows. These swirlers can be fitted with flat vanes or curved vanes. The flow fields generated by both the axial and the radial swirlers are approximately the same. Beer and Chigier [2] proposed the usage of Swirl Number, a non-dimensional parameter to characterize the amount of rotation imparted to the axial flow

\[
S_N = \frac{2 \ G_m}{D_{sw} \ G_t}
\]

where \( G_m = \text{axial flux of angular momentum} \)

\( G_t = \text{axial thrust} \)
If swirl number is less than around 0.4, no flow recirculation is obtained and the swirl is therefore described as weak. Most swirlers operate under strong swirl conditions. For swirl number greater than 0.6, the swirl is described as strong. The primary function of the swirler is to induce combustion products to flow upstream to meet and merge with the incoming fuel and air. For weak swirl there is hardly any flow recirculation, but when the swirl number is increased and reaches a critical value ($S_N > 0.4$), the static pressure in the central core just downstream of the swirler becomes low, thereby creating flow recirculation. Kilik [4] carried out velocity measurements along the swirler axis for several swirler designs and established the influence of key geometric parameters on the reverse mass flow rate. His results showed that the curved-vane swirlers induce larger reverse mass flow rate. Kilik [4] also studied the separate effects on recirculation zone-size. His experimental data show that the size of the recirculation zone is increased by increasing the vane angle, increasing the number of vanes, decreasing the vane aspect ratio and changing from flat to curved vanes.

**1.5 Focused Cooling/Backside Cooling**

To achieve present day stringent emission levels it is necessary to maintain the combustion zone temperatures within allowable limits. This essentially requires more air to be utilized in the premixing process and reaction zones. This means that the combustor liner is to be cooled without the use of cooling air bleed as hot-side film cooling. Therefore cooling must be achieved through back-side convective methods alone. Earlier, various methods for the cooling
of combustor liners have been tried and tested. In order to ensure system durability and long life, combustor liner cooling must be predictable and reliable [5].

Due to the above mentioned reasons, modern DLE combustor liners are designed to operate with minimum usage of film cooling. Complete cooling across the combustor length is just not practical, as the amount of cooling air available is limited. However, with data revealing the accurate heat load distribution on the liners, an intelligent cooling system can be designed, which focuses more of the coolant air onto the localized hot spots and prevent unnecessary coolant air from reaching the relatively cooler areas. Therefore, a highly efficient cooling system with focused cooling configurations plays a critical role in improving the efficiency of gas turbine engines and meeting the pollution emissions requirements. Solar Turbines, in their effort to reduce emissions, have introduced the Augmented Backside-Cooled (ABC) liner. Here the cooling air does not mix with the combusting mixture in the combustor primary zone. This eliminates a significant amount of quenching, which inherently lowers the CO emissions. This design also allows the combustor to be designed for a cooler flame that reduces NOx emissions.

Figure 1.9 shows a typical backside cooled combustor.

1.9: Focused Cooling [2]
1.6 Soft Wall

A softwall is basically a wall with holes in it. It is usually placed in the front wall of a combustor for the following functionalities, (1) minimize the main flame NOx emission effect due to the reduced burner air flow by the bias flow entering into the combustor primary zone, (2) minimize the effect on the turbine inlet temperature profile and (3) maximize cooling and life improvement. The introduction of a soft wall in an existing combustor design leads to changes of the cooling layout due to the convection cooled design, since some of the cooling air exits through the damping holes. In addition, the local cooling is significantly improved by the reduced heat load due to the bias flow, the cooling effect of the holes and the improved efficiency of the impingement cooling, due to reduced cross flow. Figure 1.10 shows a sample combustor showing the location of the soft wall (marked in red) and the recirculation zones [6]. The introduction of cooling bias flow significantly reduces the local heat load. The bias flow into the combustion chamber causes cooling of the hot gases in the near wall region, which influences the heat load. It also reduces the near wall convection caused by the recirculation zones. The corner regions of a combustor experience relatively large thermal gradients causing different thermal expansion of the hot liners and the relatively cold hood. Hence an improvement can be obtained with the soft wall which is beneficial for the combustor life.
1.7 Literature Survey

Older gas turbine engines used combustion dilution air and film cooling of various types as a solution to the cooling problems in combustor liners. Several studies have been conducted by Chin et al [7], Metzger et al. [8], Andrews et al. [9] and Fric et al. [10], Schulz [11] presented a first-rate review of combustor cooling by film cooling methods as well as combined film and convective cooling methods.

Ferrara et al. [12] studied convective and film cooled combustor analysis and developed an analysis tool including impingement, turbulated and film cooling methods. Smith and Fahme [13] focused on liner designs without film cooling for low emission combustors dealing with rib turbulated channel type flow cooling. Bailey et al. [5] conducted experiments and numerical simulations to understand the heat transfer characteristics of a stationary gas turbine combustor liner cooled by impingement jets and cross flow between the liner and sleeve.
Several articles that focus on the development of DLE combustors for industrial gas turbine engines have been published. Studies by Smith et al. [13, 14, and 15], Vandervort et al. [17], White et al. [18] and Roberts et al. [19] have focused on development of low NOx combustors with the viewpoint of producing lower emissions. Although the designs of these combustors vary, all of them use minimal film cooling. Arellano et al. [20] presented a study on an effective backside cooling scheme for an ultra-lean premixed combustion system. The augmented backside cooled liner, in their study, eliminates film cooling in the combustor primary zone and uses trip strip turbulators along the cold side of the liner to enhance heat transfer and thermal barrier coatings on the gas side liner wall to reduce heat load. Behrendt et al. [21] recently designed a test rig for the characterization of advanced combustor cooling concepts for gas turbine combustors. The test rig is intended to allow investigations at elevated pressures and temperatures representing realistic operating conditions of future low emission combustors. Lu et al. [22] recently studied the effect of different swirl angles for a Dry Low Emission combustor on flow and heat transfer distributions.

Patil et al. [23] studied the heat transfer distribution on a can combustor liner. They indicated that the peak value and the location of the heat transfer coefficient on the liner wall are not affected by changes in flow Reynolds number through the swirler. They also showed that the peak value with respect to the pipe flow correlation based heat transfer coefficient decreases with increasing Reynolds number. The significant reduction of the ratio of 10-12 for a Re=50,000 to around 2.5-3 for a Re=600,000 is observed. It also appears that local turbulence intensity decreases with increasing Reynolds number resulting in the observed reduction in heat transfer enhancement at higher Reynolds number. This study is a precursor to the current study and provides some valuable insight into combustor flow and heat transfer.
In an effort to achieve a fuel lean mixture, the fraction of the cooling air has to be reduced from up to 50% of the combustor air in conventional combustors to less than 30 percent in lean combustors. Hence, novel combustor cooling concepts have to be developed, which makes possible this reduction of cooling air. Even with present day advanced CFD capabilities, experimental analysis plays a crucial role in the understanding and development of combustor cooling technology. There are no published studies on experimental measurement of heat transfer distributions on the gas side liner surface for an annular combustor.

1.8 Experimental Objectives

It is clear that compared to conventional combustors, “Dry Low Emission” combustors are a better solution for compromise between emissions level and performance. The goal is to minimize film cooling and ensure that back-side wall cooling is capable of cooling the combustor wall acceptably and adequately. Due to the fact that the available coolant air is limited, focused cooling approach is required to properly allocate the cooling supply based on the local demand within a combustor. Thus it is imperative that the actual heat load distribution on the liner walls at different flow conditions is established.

This study is an effort to study the effect of flow through a high-angle swirler nozzle and understand the thermo-fluid dynamics and fundamental physics on liner heat transfer. The objectives of this study are to experimentally measure the heat transfer distribution on the liner wall in an effort to determine the locations of higher heat load. The results at various Reynolds numbers will be analyzed to deduce any possible trends. The pressure distribution along the combustor liner walls was also measured. Similarity analysis is an important step in these experiments, since they are not conducted at actual engine conditions, but at conditions suitable
for accurate testing. Dimensionless parameters are used in similarity analysis in order to relate
the low temperature results to engine conditions. Two important dimensionless parameters used
in this study are Nusselt number and Reynolds number.

Nusselt number: \( \text{Nu} = \frac{hD}{K} = \frac{q^{''\text{convective}}}{q^{''\text{conductive}}} \)

Where, \( h = \text{convective heat transfer coefficient} \)

\( D = \text{characteristic length} \)

\( K = \text{thermal conductivity of the fluid} \)

The Nusselt number is a dimensionless number and is defined as the ratio of convection heat
transfer to conduction heat transfer, where the heat conduction is under the same conditions as
the heat convection.

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

Where, \( \rho = \text{density of the fluid} \)

\( U = \text{mean fluid velocity} \)

\( D = \text{characteristic length} \)

\( \mu = (\text{absolute}) \text{ dynamic fluid viscosity} \)

In fluid mechanics and aerodynamics, the Reynolds number is a measure of the ratio of inertial
forces to viscous forces and it quantifies the relative importance of these two types of forces for
given flow conditions.
CHAPTER 2: EXPERIMENTAL SETUP

The experiment was conducted in the turbo machinery lab at Virginia Polytechnic Institute and State University. The experimental setup was scaled to a factor of two to match the original engine Reynolds number and Mach number. The total length of the setup is about 3.35 meters. There is a blower at the end of the setup to provide the required airflow. A VTAC-9 controller is attached to the motor of the blower to control the speed. The first 2.44 m of the setup acts as a settling chamber before the air goes through the swirlers. The trailing 0.91 m of the setup has holes on the sides where an IR camera lens can fit to take temperature measurements and also a heating blanket on the opposite side, this acts as the testing/combustion chamber. The setup is circumferentially made up of a quarter of an annulus; therefore it contains three swirlers in that quadrant, which is a correlation to the whole annulus of a combustor which has twelve swirlers. The test was run for both the concave and convex surface of the annular combustor surface. A blanket heater is placed in the test section to simulate a constant heat flux boundary condition. An IR camera is used to capture the temperature distribution on the heater. Thermocouples are placed at required positions to obtain the inlet temperature and to calibrate the IR camera. A pitot probe is positioned in the test setup to obtain accurate values of the velocity of air in the chamber and also to check the Reynolds number. Figure 2.1 shows a general overview of the whole setup and its component. An in depth description of all components used for the experiment will follow.
Figure 2.1: Schematic and pictures of test setup showing swirler placement (By Author)
2.1 Inlet Air Supply/Blower

The air supply to simulate air from a compressor was produced by a blower manufactured by Cincinnati Fans. The maximum capacity is about 9000 CFM depending on the static pressure. This blower is fitted with a 15 Hp motor with a maximum speed of 3525 RPM. To get the right amount of flow and Reynolds number, the motor of the blower is controlled by a VTAC 9 controller from Rockwell Automation. The controller allows the user to control the frequency of the motor which in turn determines the speed and mass flow of the air. A graph of frequency of the controller against dynamic pressure in the test section is plotted and included. The dynamic pressure can be converted to speed to determine the Reynolds number.

![Cincinnati Fans blower and VTAC 9 controller](image)

Figure 2.2: Cincinnati Fans blower and VTAC 9 controller
Figure 2.3: Graph of motor frequency against convex combustor pressure

Figure 2.4: Graph of motor frequency versus concave combustor pressure
2.2 Swirler

After the air from the blower is stabilized in the settling chamber it then goes through generic swirlers provided by solar turbine into the test chamber/combustor. A swirler is used in industries 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. 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. Due to the wakes or recirculation vortices, which are caused by turbulent flow boundary layer separation on the surface of the 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. Furthermore, the recirculation zone behind the bluff body will contribute as a flame stabilizer to help trapping the flame at high speed flow condition. Since a generic swirler is being utilized, the fuel injectors are absent but the swirlers provide recirculatory flow.

Figure 2.5: Schematic of the generic swirler produced by solar
2.3 Combustion Chamber

Air enters the combustion chamber via the swirlers. The combustion chamber section was fabricated using sheet metal, it was chosen despite the opacity because the radius required was quite large and existing transparent structures like Plexiglas and PVC pipes could not meet the size requirement without a large expense. Windows for positioning the IR camera lens were cut with equal spacing at nine different locations along the main combustion chamber for heat transfer data acquisition as shown in Figure 2.6. The windows are basically 8.25cm diameter holes. Diametrically opposite to these windows, a surface heater assembly was mounted along the combustor wall. A thick insulation material was put between the heater and the sheet metal to prevent heat loss. Each of the unused IR camera window holes were sealed during tests. A pitot probe and a five hole probe was inserted into the chamber to ascertain the various air velocity component values and give an idea of the flow profile.

Figure 2.6: Test setup showing combustion chamber/test chamber
2.4 Surface Wall Heater (constant heat flux)

In order to obtain constant heat flux boundary conditions at the liner wall, the combustor wall diametrically opposite the IR windows was fitted with a heater from Omega. Two different heater sizes were used to fit the side of the combustor being analyzed i.e. concave or convex. Figure 2.7 shows the schematic of the surface heater system construction and energy processes for the steady-state experiment. The face of the heater to be analyzed was coated with a thin film of black paint for enhanced emissivity. The back of the heater is the glued to an insulation material that is attached to the inside of the combustor/test section. Heat generated from the heater was adjusted through the use of transformers by varying the voltage and amperage to obtain the required power output.

The entire outside wall of the combustor liner model was insulated using a thick layer of rubber insulation. However, a small amount of heat loss from the heater, by conduction into the liner wall and by radiation to the surroundings was unavoidable. The amount of heat loss is a function
of the temperature of the combustor liner wall. The temperature of the liner varies along the liner length. The temperature is lower just downstream of the swirler where the swirl motion increases convective heat transfer and it is higher further downstream where the flow mixes back into chamber. Due to this temperature difference the heat loss along the liner will vary with respect to temperature. In order to establish this loss, the liner wall was heated without supplying air into the combustor model. Therefore all the energy generated will be dissipated via conduction and radiation and not by forced convection. In other words the heat energy dissipated in this case will be the heat loss when air is blown through the combustor. The heat loss occurring at varying temperatures is plotted below.

![Graph showing heat loss from liner wall as a function of temperature](image)

Figure 2.7a Heat loss from liner wall as a function of temperature.
2.5 Infrared Thermal Imaging System

The FLIR SC640 Infrared Camera, shown in Figure 2.8 was used to capture the liner wall surface temperature distribution within the combustor. The SC640 camera is a focal plane array system type IR camera using micro bolometer as detector material and has thermal sensitivity as high as within 0.1°C at 30°C. The camera has a maximum resolution of 640x480 and wide measurement range of -40°C to +1,500°C, in 3 ranges. With proper calibration temperatures up to + 2000°C can be recorded. 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. Thermography is the production of temperature calibrated infrared or heat pictures by utilizing an infrared camera. Based on these thermal images, accurate temperature measurements can be made to detect even the smallest temperature differences. It is not always possible to know where to attach the thermocouples necessary to make accurate measurements and effectively evaluate heat dissipation. Furthermore, since the thermocouple needs to be in contact with the component to be tested, it can influence the results of the measurement. The ThermaCAM SC640 has the advantage that it produces very comprehensive images in a non-contact mode. It can produce very high resolution images (640 x 480 pixels) so that crisp thermal images can be taken of even the smallest of objects in a non-contact mode. The camera is especially designed for the most demanding scientific applications and detects the smallest temperature differences over a very wide temperature range. Frames can be captured, and stored, in real-time, at high frame rates allowing for detailed and extensive analysis of highly dynamic events typically found in R&D environments.
Figure 2.8: FLIR SC640 Infrared Camera

Figure 2.9: Sample combustor image taken by IR Camera
2.6 Personal Daq USB Data Acquisition Module

Various thermocouples were positioned to obtain temperature values at inlet to the combustor and also to calibrate the IR camera. The OMB-DAQ-54 Personal Daq, as shown in Fig 2.10 was used to acquire the thermocouples process signals. It is a full-featured data acquisition product that uses the Universal Serial Bus (USB) built into almost every new PC. Designed for high accuracy and resolution, the OMB-DAQ-54 data acquisition systems directly measure multiple channels of voltage, thermocouple, pulse, frequency, and digital I/O. A single cable to the PC provides high-speed operation and power to the OMB-DAQ-54. No additional batteries or power supplies are required, except when using bus-powered hubs. Because of the strict power limitations of the USB, the modules incorporate special power management circuitry to ensure adherence to USB specifications. The OMB-DAQ-54 module avoids many of the limitations of PC-card (PCMCIA) data acquisition devices and offer advantages over many PC plug-in data acquisition boards as well. The OMB-DAQ-54 data acquisition system offers 10 single-ended or 5 differential analog (up to ±20 V full scale) or thermocouple input channels with 16 programmable ranges and 500 V optical isolation. Two type K thermocouples were positioned at the inlet to the combustor to estimate the inlet temperature values. One cement-on thermocouple was positioned on the combustor wall opposite one of the IR windows to calibrate the IR camera.
Figure 2.10: OMB-DAQ-54 Personal Daq

2.7 NetScanner 9816/98RK Rackmount Ethernet Intelligent Pressure Scanner

The NetScanner System is a multi-channel pressure acquisition. The system is comprised of a Model 98RK Scanner Interface Rack, housing up to eight Model 9816 Intelligent Pressure Scanners networked via the Ethernet interface. NetScanner System Intelligent Pressure Scanner modules are flexible pressure measuring devices intended for use in research and production environments. A single model 9816 Intelligent Pressure Scanner is available with sixteen measurement channels, each with individual pneumatic transducers per channel. During the experiments five of the channels were used in conjunction with a five hole probe to capture the pressure distribution in the combustion chamber due to the swirler. Model 9816 pressure scanners are capable of accuracies up to ±0.05%. Accuracy is maintained through use of built-in
re-zero and span calibration capabilities. 98RK Scanner Interface Rack features pneumatic hook-ups on the back-panel or front-panel (if ordered) to ease calibration of the scanners. Each 9816 Intelligent Pressure Scanner module contains a pneumatic calibration manifold and software commands to automatically perform re-zero and span adjustment calibrations. The 98RK chassis requires an 80 psig minimum dry air (or inert gas) supply which is used to shift the 9816 internal calibration valve (in each scanner) between its different positions. Each internal Model 9816 pressure scanner module (mounted in 98RK chassis slots) has an Ethernet interface to communicate with a host computer.

Figure 2.11 NetScanner multi-channel pressure acquisition system
CHAPTER 3: EXPERIMENTAL METHODOLOGY

The core temperature of a combustor can be as high as 1850K which is about the melting point temperature of the materials used to design the combustor. The operating pressure within a combustor can be around 20 atm to 30 atm. It will be difficult to duplicate these conditions in our test facilities. Capital costs and safety issues will not make it feasible. A non-reactive combustion investigation approach was considered as a more realistic and feasible option for this project in accordance with our available resources. The intake air temperature was at room temperature and the operating pressure was the local atmospheric pressure. The experiments were conducted at various Reynolds numbers (210000, 420000 and 840000). To correlate the experimental results to real time engine conditions values, it is imperative to conduct similarity analysis to establish the relationship between actual heat transfer coefficient and test model heat transfer coefficient.

The experimental apparatus was set up on a one to two scale.

<table>
<thead>
<tr>
<th>Real Engine Conditions:</th>
<th>Test Model Conditions:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air Temp. = 1850K</td>
<td>Air Temp. = 293K</td>
</tr>
<tr>
<td>$k_{\text{air}}@1850 = 0.124 \text{ W/m K}$</td>
<td>$k_{\text{air}}@293 = 0.0263 \text{ W/m K}$</td>
</tr>
<tr>
<td>Operating pressure = 20 atm</td>
<td>Operating pressure = 1 atm</td>
</tr>
</tbody>
</table>

Similarity Analysis:

\[
(Nu)_{\text{actual}} = (Nu)_{\text{test}}
\]

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

\[
h_{\text{actual}} = \left(\frac{k_{\text{actual}}}{k_{\text{test}}}\right) \left(\frac{D_{\text{test}}}{D_{\text{actual}}}\right) h_{\text{test}} = 4.71 \times 2 \times h_{\text{test}} = 9.42 \times h_{\text{test}}
\]
Therefore it is evident that the actual heat transfer coefficient at real engine condition should be about 9.42 times higher than the experimental results. It is imperative to conduct similarity analysis so that the experimental results can be utilized to validate the real engine operation investigation.

The air supply was provided by a 20-hp high pressure blower whose motor was controlled by a V-Tac 9 variable frequency controller to obtain the set Reynolds number. This centerline velocity, measured by the pitot probe, was adjusted to obtain the required Reynolds number. The air was passed into the combustor simulator section through three swirlers fitted at its entry. The annular section represents a quadrant of the actual full annulus of the engine combustor. Each swirler was a scaled up prototype of the engine scale swirler and provided by Solar Turbines. The air enters an annular section upstream of the swirlers and then enters the actual combustor section through the three swirlers producing a type engine combustor type flow without combustor. The walls of annulus, only the convex and concave are heated using thin wall heaters to produce a constant heat flux surface. An infrared (IR) camera was used to measure the surface temperature on the combustor liner during a steady-state experiment. The typical combustor nozzle has a 10-vane swirler. The outer diameter of the swirler is about 14.5cm and the inner diameter is about 7.98cm, therefore the effective flow area is about 114.2cm². The flow then enters the annular combustor model which is a quadrant of a whole annulus with a hydraulic diameter of 0.7m. The combustor test section is 0.91m in length. The air flow is set up by measuring the centerline velocity upstream of the swirlers with a pitot probe. A fine gauge T-type thermocouple is used in conjunction with an OMB-DAQ-54 Personal Daq USB Data Acquisition Module to verify the temperature of the free stream air. The blanket heater is turned
on simultaneously with the blower. The whole setup takes about an hour to attain steady state condition in the test chamber.

![Figure 3.1 Test section configuration with three swirlers](image)

Once the setup is ready for temperature data acquisition, the IR camera is calibrated with in-situ thermocouple placement. The surface of the heater is painted black to obtain a surface emissivity of around 0.95. The IR camera is placed with the lens fitting into the observation holes diametrically opposite the heater and the whole setup is covered with a black blanket, this is done to prevent any external light source from interfering with our data acquisition. Since there are no IR windows, there is no necessity to consider window transmissivity. To help calibrate the IR camera a few K- type thermocouples are attached to the heater with conductive cement and connected to the personal DAQ system. The emissivity on the IR camera is altered on a 0-1 range until the IR temperature reading on the thermocouple spot matches the temperature reading of the thermocouple. The IR camera is used to capture the thermal picture of the section being analyzed. The thermal pictures from the different port holes are merged together and analyzed to
determine the temperature distribution on the liner. The basic convective heat transfer equation was then used to convert the wall temperatures to heat transfer coefficients as shown below:

\[ h = \frac{Q}{A(T_{wall} - T_{air})} \]  

(3.1)

The wall heat flux was calculated using the resistance ratings of the heater and the voltage settings on the transformer.

\[ Q = \frac{\nu^2}{R} \]  

(3.2)

The local Nusselt number \((Nu = \frac{hD_h}{k})\) is based on the annulus hydraulic diameter. For flow distribution measurements, a pitot tube and was inserted through the observation holes to acquire local pressure data.

For further heat transfer and flow analysis, a softwall is introduced on the front panel of the existing annular combustor. Different sizes and configurations of the softwall are analyzed to deduce the effect on the heat transfer. Figure 3.3 shows the location of the softwall zones. The softwall was placed circumferentially around the middle swirler and located midway between the swirler and the liner surface. The number and size of the holes were varied for different runs of the experiment.
Figure 3.2: Schematic of concave and convex test sections

Figure 3.3: Combustor front panel with softwall
3.1 Uncertainty Analysis

An uncertainty analysis was performed for the equations used as well as for the boundary conditions; this should provide an overall confidence in the experiment. First a conservative error estimate of the measured quantities was determined and then the relative uncertainties in measured quantities were calculated. Finally, the overall average uncertainty was calculated by taking the square root of the summation of the square of all the relative uncertainties.

Wall heat flux uncertainty analysis

For the constant wall heat flux, the resistance rating on the heater was used to deduce the value based on the equation 3.2. \( Q = \frac{V^2}{R} \)

Error estimate for the variables are as follows:
\( \Delta R = 0.45 \Omega \) (obtained from the ratings for the heater)
\( \Delta V = 2V \) (LC of the variable transformer used to set the voltage during the experiment)

The relative uncertainties of the variables are:
\( U_R = \Delta R/R \)
\( U_V = \Delta V/V \)

The root mean square uncertainty in calculating the heat flux:
\( U_Q = \sqrt{(2U_V)^2 + (U_R)^2} = \pm 5.1\% \)

Heat transfer coefficient uncertainty analysis

Heat transfer coefficient for flow in the combustor model was calculated using equation 3.1.
\[ h = \frac{Q}{A(T_{wall} - T_{air})} \]
Error estimate for the variables are as follows:

\[ A = L \cdot B \]
\[ \Delta B / \Delta L = 0.0625 \text{in} \]
\[ \Delta T_{\text{wall}} = 0.5^0 \text{C} \]
\[ \Delta T_{\text{air}} = 0.5^0 \text{C} \]

The relative uncertainties of the variables are:

\[ U_Q = \pm 5.1\% \]
\[ U_L = \Delta L / L \]
\[ U_B = \Delta B / B \]
\[ U_{T_{\text{wall}}} = \Delta T_{\text{wall}} / T_{\text{wall}} \]
\[ U_{T_{\text{air}}} = \Delta T_{\text{wall}} / T_{\text{wall}} \]

The root mean square uncertainty in calculating the heat transfer coefficient:

\[
U_h = \sqrt{(U_Q)^2 + (U_B)^2 + (U_L)^2 + (U_{T_{\text{wall}}})^2 + (U_{T_{\text{air}}})^2} = \pm 5.1\%
\]

**Reynolds number uncertainty analysis**

\[ \text{Re} = (V \cdot D) / \nu \]

Error estimate for the variables are as follows:

\[ \Delta V = 0.1 \text{ft/s} \]
\[ \Delta D = 0.0625 \text{in} \]

Here, \( \nu \) is a tabulated value. The rule of thumb is to assume 3\% relative uncertainty

\[ U_V = 0.03 \]

The root mean square uncertainty in calculating the Reynolds number:
U_h = \sqrt{(U_p)^2 + (U_D)^2 + (U_D)^2} = \pm 4.0\%

**Nusselt number uncertainty analysis**

\[ \text{Nu} = \frac{(h \times D)}{k} \]

Error estimate for the variables are as follows:

\[ \Delta D = 0.0625\text{in} \]

The relative uncertainties of the variables are:

\[ U_k = 0.03 \text{ (rule of thumb)} \]

\[ U_h = 5.7\% \]

The root mean square uncertainty in calculating the heat flux:

\[ U_{Nu} = \sqrt{(U_h)^2 + (U_D)^2 + (U_k)^2} = \pm 6.4\% \]

Therefore, the overall average uncertainty in Nu for the experiment was estimated to be \( \pm 6.4\%\).
CHAPTER 4: RESULTS

The experiment was run by pushing air through the blower into the test section. The Reynolds number is checked and adjusted as needed. The IR camera measured the surface temperature of the combustor. The temperature values were then used in a MATLAB program to calculate the heat transfer coefficient values and other thermal properties. Thermocouples placed downstream of the swirler provided the ambient intake air temperature. The constant heat flux was calculated by using the resistance ratings of the wall heater and the output voltage set on the transformer. The obtained power (wattage) was then divided by the area of the heater to obtain the constant heat flux value.

The local measured Nusselt number was normalized using the Nusselt number prediction for a pipe flow with the typical pipe flow Reynolds number. Heat transfer coefficient at the liner wall is characterized by the Nusselt augmentation ratio, where the baseline Nusselt number is obtained from the Dittus-Boelter correlation for fully-developed flow

\[ Nu_{fd} = 0.023 Re_D^{0.8} Pr^{0.3} \]  \hspace{1cm} (4.1)

With the help of CFD and experimental measurements, the flow characteristics in the test chamber were mapped [25]. The contours of the axial velocity profiles upstream and downstream are presented as well as the experimental axial flow contours. Some insight is also provided on the other velocity components and the effect of the swirl number. Effect of the softwall introduction on the liner heat transfer is also included in the results.
4.1 Thermal Analysis

The IR camera is used to capture 2D thermal images of the combustor liner; the images are then analyzed and plotted in Matlab. Figure 4.1 and 4.2 show sample images used for the test.

Figure 4.1 Sample IR image of liner

Figure 4.2 Processed image of IR picture
Measurements were performed on both the concave and convex surfaces of the annular combustor model. The results are then compared to determine any variations in the heat load on both surfaces.

Figure 4.3 shows the surface temperatures and the calculated heat transfer coefficient at steady state for a concave surface. The set experimental conditions of a constant wall heat flux of 774.5 W/m\(^2\) with a free-stream temperature was used to obtain the local wall temperatures. The Reynolds number through the test section was 420,000. It is clear that the cooler locations produce the higher heat transfer coefficients. The flow exits the swirler and impinges on the concave wall a little further downstream creating the local high heat transfer region. Further downstream the impinging flow is deflected back to the middle of the channel and heat transfer coefficient decrease with increasing distance from the swirler. This trend is similar to the study by Patil et al. [23] on can combustor liner walls.

![Wall temperature and heat transfer coefficient](image)

Figure 4.3 2-D wall temperature and heat transfer coefficient (W/m\(^2\)-K) distributions on the concave liner wall at Re=420000
Figure 4.4 shows the wall temperature and heat transfer coefficients for the convex wall. It is clear that the trends are similar to concave wall. However, the high heat transfer coefficient region is lower and spreads wider than for the concave wall. The convex wall may cause more diffusive flow near the wall due to the nature of the curvature.
Figure 4.5 Comparing convex and concave surface heat transfer enhancement along the combustor liner

Figure 4.5 shows the comparison of the concave and convex surface heat transfer coefficients at Re=420,000. It is clear that the peak region heat transfer coefficients are higher for the concave surface although the peak location is similar. The concave surface heat transfer coefficients continue to be slightly higher than the convex surface values through the entire liner surface.

Figure 4.6 shows the effect of flow Reynolds number on heat transfer augmentation on the concave surface. As it can be seen from Figure 4.6, the trends were consistent for all the
Reynolds numbers but, the peak value of the augmentation reduced as the Reynolds number increased. The location of the peak value does not change with Reynolds number. This is consistent with observations of Patil et al. [23] in the can combustor study.

Figure 4.6 Effect of Reynolds number on Nusselt number augmentation for concave surface
Figure 4.7 shows the effect of Reynolds number on the convex surface. The trends are similar to that for the concave surface. However, the drop in heat transfer augmentation from 420,000 to 210,000 is not as significant as for the concave surface.
4.2 Validation with Computational Model

Figure 4.8 and Figure 4.9 compare predictions with experiment for Reynolds number of 420,000. The results are plotted versus normalized axial distance along the combustor liner wall. It is observed that predictions compare very well with experiments. The computational model predicted both the location and magnitude of peak heat transfer in very good agreement with experiments for both the convex and concave liner walls. However, some difference between the model prediction and experiments exist downstream of the peak location. The computational length of the combustor was much smaller than that used in the experiments to limit the mesh size. The proximity of the outflow condition could possibly affect the decay of the augmentation ratio. However, the same trends between prediction and experiments were observed in an earlier study with a can combustor [23].

![Figure 4.8 Nusselt number augmentation on concave liner wall](image)
Figure 4.9 Nusselt number augmentation on convex liner wall
4.3 Flow Field Measurements

A pitot tube was used to map the axial velocity profile in the combustor both in the axial direction and the transverse section from convex and concave surface. Figure 4.10 shows the 2-D profile of the axial velocity downstream of the swirler. It clearly shows that there is an axial velocity deficit just downstream of the swirler as the flow moves away from the center of the combustor towards the walls. Further downstream, the velocity is more uniform across the span of the combustor as the impinging flow mixes back into the core region.

Figure 4.10 2-D axial velocity profile in the annular combustor
The axial velocity was averaged horizontally across the cross section of the annulus and
displayed in Figure 4.11, and the axial velocity was averaged along the length of the test section
and is presented in Figure 4.13. The general flow characteristic stay the same for all the
Reynolds numbers but the magnitudes are higher as the Reynolds number increases.

![SECTION A-A](image)

Figure 4.11 Transverse section of axial flow profile

It can be deduced from Figure 4.11 that the concave and convex flow properties are quite similar
with a few variations. The velocity near the concave wall is slightly higher than the velocity
profile closer to the convex surface which tends to also have a wider span of the peak velocity
profile than the concave surface. This could be due to the curvature of the convex surface which induces a diffusive flow in that region. The velocity seems to increase as the flow moves from the swirler towards the liner walls; this can be attributed to the corner recirculation regions induced by the swirler. Figure 4.12 shows the streamline contours produced by the numerical model. A close look at the profile shows more flow recirculation at the concave surface. It also shows the corner recirculation zones and also a slight central recirculation zone, this compliments and validates the experimental flow results in Figure 4.11. The presence of the central recirculation zone indicates that the swirl number is beyond the critical value of 0.6. The swirl number is defined as the ratio of tangential momentum to axial momentum and is calculated as

\[
S = \frac{\int r V_\theta V_x \, dr}{R_0 \int V_x^2 \, dr}
\]

(4.2)

at an axial plane near the swirler nozzle exit was found to be 0.98
Figure 4.12 Contours streamlines in computational domain [24]
Figure 4.13 shows a general decay in axial velocity as the flow progresses downstream. This can be attributed to the high swirl upstream, but the swirled flow mixes back into the core of the combustor which results in the gradual reduction in the axial velocity downstream. Figure 4.14 by Patil shows a similar trend in which the axial velocity is highlighted at the swirler exit but decays downstream.
4.4 Comparison of Heat transfer and Flow Results

It can be deduced from the previous results that the peak heat transfer region is unaffected by Reynolds number. Now the thermal and flow results will be compared to check for any trends that might be present. As can be seen from Figure 4.15 below; when the axial velocity profile along the liner is superimposed with the Nusselts augmentation along the liner, the peak location align themselves close together and the general trend of high turbulence activity upstream and decay downstream also matches.
There was a slight variation in the heat transfer profile on the concave and convex surface. The concave surface had a higher heat transfer peak which followed throughout the combustor length. The convex heat transfer had a wider span at the peak location which is attributed to the curvature of the surface. Comparing these effects to the turbulent kinetic energy contours provided by Patil [24] in Figure 4.16 below shows the variation on the two surfaces.
Figure 4.16 Contours of normalized turbulent kinetic energy in meridional plane in computational domain [24]

The convex wall shows a wider span of not so energetic flow and the concave surface shows a narrower band but more energetic flow. This is what results in the concave surface being slightly higher than the convex surface. The result is also concurrent with the A-A section of the experimental flow results shown in Figure 4.11

4.5 Comparison of Combustor Augmentation

Figure 4.17 compares the peak value heat transfer coefficient enhancement with Reynolds number effect for both the concave and convex surfaces with the can combustor results [23]. It appears that at lower Reynolds number, the peak value studied for the can combustor shows significantly high enhancement at around 10-12 and drops rapidly with increasing
Reynolds number to about 2 at Re=500,000. This was attributed by Patil et al. [23] to the drastic reduction in turbulence intensity at higher Reynolds numbers. For annular combustors, both convex and concave surfaces do not show a significant drop at higher Reynolds numbers, the peak values drop from 6 to around 4 for increase in Reynolds number from 210,000 to 840,000. This indicates that there are significant differences in the way the flow develops after the swirler for can and annular combustors and the nature of heat transfer enhancement depends on it.

Figure 4.17 Variation of peak heat transfer augmentation with Reynolds number
4.6 Soft Wall Results

As shown in Figure 4.18 below the soft wall generally reduces the swirl impingement on the liner produced by the swirler which in turn reduces the peak heat transfer on the liner wall. This reduction is highlighted when the size of the soft wall holes increase and when the number of holes increases. As stated earlier the bias flow cools the liner and increases convection in the combustor. In effect, it is a situation whereby the combustor liner is being cooled from both sides. Comparing the two extremes, i.e. the liner with the most and bigger holes for the softwall and the liner with no softwall, it is apparent that the peak heat transfer location decrease with the softwall and the general profile heat transfer is lower in the softwall case. This is detailed in Figure 4.19 which compares the heat transfer coefficient profiles on the liners with softwall and the one without it. This is an interesting observation that warrants more studies in the future, it will be a compromise between allocating air for backside cooling and the softwall, and whether the reduction in the already diminishing coolant air will have adverse effects on the liner cooling or whether the softwall bias flow will compensate for that.
Figure 4.18 Effect of soft wall on the local peak heat transfer.
Figure 4.19 Comparison of heat transfer coefficient of liner with and without softwall
CHAPTER 5: CONCLUSIONS

Most present day DLE combustors reduce the usage of film cooling and have resorted to back-side liner cooling in an effort to reduce NOx, CO and UHF emissions which result from localized hot and cold spots. The goal of this project was to study the heat transfer distribution and flow characteristics inside a “Dry Low NOx Emission” combustor model equipped with a swirler, provided by Solar Turbines, Inc., in order to better understand the flow behavior that affect the heat loads within a turbine combustor to help improve the current cooling system, and design a more effective cooling system. The overall aim of this study is to provide a better understanding of the combustor swirling flow and its effect on liner surface heat transfer. Detailed heat transfer coefficient measurements on the liner wall using Infra-red thermography were presented at different Reynolds numbers.

Detailed heat transfer and flow distributions were measured and predicted in scaled up annular combustor models. The convective heat transfer coefficient distributions on the liner walls are critical in designing efficient cooling schemes for modern DLE combustors. The highly swirling flow coming out of the swirlers impinge on the liner walls and enhance heat transfer coefficients significantly. The effect of Reynolds number was investigated and the results indicate that the heat transfer augmentation compared to a pipe flow fully developed flow condition decreases at higher Reynolds number but the peak location for enhancement is unaffected.

The CFD simulations accurately capture the trend in heat transfer coefficients and provide interesting detail into the swirling flow behavior coming out of the swirler. The RNG model used in the CFD predictions performs well in capturing the flow and heat transfer behavior in such a complex geometry. There are some interesting differences between the convex and concave
surfaces with the concave surface showing higher heat transfer coefficients than the convex surface. The degradation in heat transfer enhancement in annular combustors at higher Reynolds number is not as significant as for the can combustor geometry studied by Patil et al. [21]. The introduction of a softwall in the combustor reduces the local peak heat transfer on the liner.
CHAPTER 6: REFERENCES


