CRANFIELD UNIVERSITY

Bhupendra Khandelwal

DEVELOPMENT OF GAS TURBINE COMBUSTOR PRELIMINARY DESIGN METHODOLOGIES AND PRELIMINARY ASSESSMENTS OF ADVANCED LOW EMISSION COMBUSTOR CONCEPTS

SCHOOL OF ENGINEERING
PhD Thesis

Supervisor: Prof. Riti Singh & Dr. Vishal Sethi

July 2012

CRANFIELD UNIVERSITY

SCHOOL OF ENGINEERING
PhD Thesis

PhD

Bhupendra Khandelwal

DEVELOPMENT OF GAS TURBINE COMBUSTOR PRELIMINARY DESIGN METHODOLOGIES AND PRELIMINARY ASSESSMENTS OF ADVANCED LOW EMISSION COMBUSTOR CONCEPTS

Supervisor: Prof. Riti Singh & Dr. Vishal Sethi

July 2012

This thesis is submitted to fulfilment of the requirements for the degree of PhD

EXECUTIVE SUMMARY

It is widely accepted that climate change is a very serious environmental concern. Levels of carbon dioxide (CO_2) and other emissions in the global atmosphere have increased substantially since the industrial revolution and now increasing faster than ever before. There is a thought that this has already led to dangerous warming in the Earth's atmosphere and relevant changes around. Emissions legislations are going to be stringent as the years will pass. Hydro carbon fuel cost is also increasing substantially; more over this is non-renewable source of energy.

There is an urgent need for novel combustor technologies for reducing emission as well as exploring alternative renewable fuels without effecting combustor performance. Development of novel combustors needs comprehensive understanding of conventional combustors. The design and development of gas turbine combustors is a crucial but uncertain part of an engine development process. At present, the design process relies upon a wealth of experimental data and correlations. Some major engine manufacturers have addressed the above problem by developing computer programs based on tests and empirical data to assist combustor designers, but such programs are proprietary. There is a need of developing design methodologies for combustors which would lead to substantial contribution to knowledge in field of combustors. Developed design methodologies would be useful for researchers for preliminary design assessments of a gas turbine combustor.

In this study, step by step design methodologies of dual annular radial and axial combustor, triple annular combustor and reverse flow combustor have been developed. Design methodologies developed could be used to carry out preliminary design along with performance analysis for conventional combustion chambers. In this study the author has also proposed and undertaken preliminary studies of some novel combustor concepts.

A novel concept of a dilution zone less combustor has been proposed in this study. According to this concept dilution air would be introduced through nozzle

guide vanes to provide an optimum temperature traverse for turbine blades. Preliminary study on novel dilution zone less combustor predicts that the length of this combustor would be shorter compared to conventional case, resulting in reduced weight, fuel burn and vibrations. Reduced fuel burn eventually leads to lower emissions.

Another novel concept of combustor with hydrogen synthesis from kerosene reformation has been proposed and a preliminary studies has been undertaken in this work. Addition of hydrogen as an additive in gas turbine combustor shows large benefits to the performance of gas turbine engines in addition to reduction in NO_x levels. The novel combustor would have two stages, combustion of ~5% of the hydrocarbon fuel would occur in the first stage at higher equivalence ratios in the presence of a catalyst, which would eventually lead to the formation of hydrogen rich flue gases. In the subsequent stage the hydrogen rich flue gases from the first stage would act as an additive to combustion of the hydrocarbon fuel. It has been preliminary estimated that the mixture of the hydrocarbon fuel and air could subsequently be burned at much lower equivalence ratios than conventional cases, giving better temperature profiles, flame stability limits and lower NO_x emissions.

The effect of different geometrical parameters on the performance of vortex controlled hybrid diffuser has also been studied. It has been predicted that vortex chamber in vortex controlled hybrid diffuser does not play any role in altering the performance of diffuser.

The overall contribution to knowledge of this study is development of combustor preliminary design methodologies with different variants. The other contribution to knowledge is related to novel combustors with a capability to produce low emissions. Study on novel combustor and diffuser has yielded application of two patent applications with several other publications which has resulted in a contribution to knowledge. A list of research articles, two patents, awards and achievements are presented in Appendix C.

Keywords: Combustor, Gas Turbine Combustor Design, Novel Combustors, Hydrogen synthesis, kerosene reforming and Hydrogen enriched combustion.

ACKNOWLEDGEMENTS

This thesis is the end of my journey in obtaining my Ph.D. I have not traveled in a vacuum in this journey. This thesis has been kept on track and been seen through to completion with the support and encouragement of numerous people including my well-wishers, my friends, colleagues and various institutions. At the end of my thesis I would like to thank all those people who made this thesis possible and an unforgettable experience for me. At the end of my thesis, it is a pleasant task to express my thanks to all those who contributed in many ways to the success of this study and made it an unforgettable experience for me.

My first debt of gratitude must go to my advisor, Emeritus Prof. Riti Singh. He patiently provided the vision, encouragement and advice necessary for me to proceed through the doctoral program and complete my dissertation. I want to thank him for unflagging encouragement, strong and supportive adviser to me. He has always given me great freedom to pursue independent work.

I would also like to give sincere thanks to my co-supervisor Dr. Vishal Sethi who has helped me right from scratch till the end. His encouragement and motivation made me feel confident to fulfil my desire and to overcome every difficulty I encountered. From very first stage to last stage of my Ph.D., he has helped me technically, corrected grammar mistakes, suggested possible improvements, and facilitated in completing any kind of forms or any advice which I needed. He went out of the way for providing every support to me.

I also appreciate the advice of Prof. Hoi Yeung, Dr. Panagiotis Laskaridis and Prof. J. N. Murthy for their critical comments, which enabled me to notice the weakness of my dissertation and make the necessary improvements according to their comments.

My sincere thanks go to all the organizations, which gave me their indispensable generous sponsoring, including the CLEANSKY, Royal Aeronautical Society and Institute of Mechanical Engineers. Without their support and financial help, it would not have been possible for me to pursue and to complete this Ph.D. project successfully.

I am very grateful for my parents Raghunandan and Preeti. Their understanding and love encouraged me to work hard and to continue pursuing a Ph.D. project abroad. Their firm and kind-hearted personality has affected me to be steadfast and never bend to difficulty. They have always lets me know that they are proud of me, which motivates me to work harder and do my best.

Last but not least, I would like to thank my lovely wife Trupti for her love, inspiration and continuous support during all the years. I would say she is the reason for which, I did this Ph.D. She has shown her patience and affection, which she demonstrated in the last and most difficult months of this thesis. Her love and support without any complaint or regret has enabled me to complete this Ph.D. project.

During my research, many MSc students have helped me in completing this project. I would like to thank all of them for their efforts. Specially. Adam Karakurt who has helped me substantially in doing bits and pieces. Our conversations enlightened my way of thinking, and so I'd like to give my sincere thanks to MSc students for their help. In the end I would like to thank Prof. Sudarshan Kumar for encouraging me to do collaborative research and being my unofficial mentor.

"The larger the island of knowledge, the longer the shoreline of mystery"….

TABLE OF CONTENTS

EXECUTIVE SUMMARY i
ACKNOWLEDGEMENTS v
NOMENCLATURE ix
1 INTRODUCTION 15
1.1 Thesis structure 16
1.2 Environmental Concerns in Aviation Industry 17
1.3 Availability of Fossil Fuel Resources 18
1.4 Conventional Combustors 19
1.5 Types of Combustors 20
1.5.1 Can or tubular combustion chambers 21
1.5.2 Can annular combustor 21
1.5.3 Annular combustors 22
1.6 Parts of Combustors 23
1.6.1 Diffuser 23
1.6.2 Fuel Injector 25
1.6.3 Liner 28
1.6.4 Air Casing 28
1.6.5 Combustion Zones 28
1.6.6 Swirler 29
1.7 Low Emission Combustors 31
1.7.1 Wet methods for emission control 34
1.7.2 Dry Low NO_x Combustors 35
1.8 Objectives and Scope 41
1.9 Contribution to knowledge and technology transfer 42
2 CONVENTIONAL COMBUSTORS - DESIGN 43
2.1 Abstract 43
2.2 Introduction 44
2.3 General Combustor Design Methodology 46
2.4 Performance Parameters 53
2.4.1 Fuel Injector Performance 53
2.4.2 Swirler Performance 55
2.4.3 Combustion Efficiency 56
2.4.4 Pattern Factor 56
2.4.5 NO_x prediction 58
2.4.6 CO prediction 59
2.4.7 UHC Prediction 60
2.5 Design Methodology for Combustors 60
2.5.1 Single Annular Combustor 61
2.5.2 Reverse Flow Combustor 65
2.5.3 Radially Staged Dual Annular Combustor 70
2.5.4 Axially Staged Dual Annular Combustor 74
2.5.5 Radially Staged Triple Annular Combustor 75
2.6 Conclusions 77
3 DILUTION ZONE LESS NOVEL COMBUSTOR 78
3.1 Abstract 78

3.2 Introduction 78
3.3 Methodology 85
3.4 Calculation for NGV 88
3.5 Discussion on Patent Search 92
3.5.1 Gas turbine engine combustion equipment 92
3.5.2 Turbine Structure – Combustion structure 93
3.5.3 Cooled Vanes 95
3.5.4 Dilution Pole combustor and method: 97
3.6 Advantages 98
3.7 Conclusions 99
4 NOVEL COMBUSTOR WITH HYDROGEN SYNTHESIS FROM KEROSENE REFORMATION 100
4.1 Abstract 100
4.2 Introduction 100
4.3 Methodology 105
4.4 Hydrogen Production methods 107
4.4.1 Hydrogen Production without catalyst 111
4.4.2 Effect of hydrogen enrichment on hydrocarbon combustion 111
4.5 Summary 117
4.6 Conclusions 118
5 CONCLUSIONS & RECOMMENDATIONS 119
5.1 CONCLUSIONS 119
5.2 RECOMMENDATIONS 121
APPENDICES 122
Appendix A Combustor Design 122
Appendix B Novel Hybrid Diffuser 194
Appendix C Publications, Patents and Awards 211
REFERENCES 221

NOMENCLATURE

A	Area
$A_{inner\_pn,eff}$	Effective area of inner atomizer passage
$A_{outer\_pn,eff}$	Effective area of outer atomizer passage
A_{inner_pass}	The inner annular passage area
$A_{sw\_pri,phy}$	Physical flow area of each primary swirler
$A_{outer\_dil\_hole,phy}$	Physical area of outer dilution holes
$A_{inner\_dil\_hole,phy}$	Physical area of inner dilution holes
$A_{inner\_pri\_hole,eff}$	Effective area of inner primary hole
$A_{outer\_pri\_hole,eff}$	Effective area of outer primary hole
A_{phy}	Physical area of primary holes
A_{outer_turn}	Surface area of outer turn section
A_{ref}	Reference area
$A_{sw\_\sec,phy}$	Physical flow area of each secondary swirler
$A_{sw\_pri\_hole}$	Area of each ellipsoidal hole
$A_{sw\_pri,phy}$	Physical flow area of each primary swirler
$A_{sw\_pri,eff}$	Effective area of each primary swirler
$A_{sw\_\sec,phy}$	Total physical area of each secondary swirler
$A_{sw\_\sec,eff}$	Total effective area of each secondary swirler
A_{shiled}	Surface area of heat shield plate
AFR	Air to fuel ratio
AR	Area Ratio
B	Axial width of vane passages

BPR	Bypass ratio
$C_{d,\infty}$	Discharge coefficient when pressure loss factor is infinite
$C_{d,outer\_dil\_hole}$	Discharge coefficient of outer dilution hole
C_{d,sw_pri}	Primary swirler discharge coefficient
$C_{d,inner\_turn}$	Discharge coefficient of inner turn section
$C_{d,outer\_turn}$	Discharge coefficient of outer turn section
$C_{d,sw\_\sec}$	Secondary swirler discharge coefficient
$C_{d,inner\_pri\_hole}$	Discharge coefficient of inner liner primary holes
$C_{d,outer\_pri\_hole}$	Discharge coefficient of outer liner primary holes
CO	Carbon monoxide
Cp	Pressure Recovery Coefficient
C_{pi}	Ideal pressure recovery coefficient
D	Diameter of dilution holes
DP	Design Point
D_{odif}	Outer diameter of diffuser inlet
D_{idif}	Inner diameter of diffuser inlet
D_{ot}	Outer diameter of turbine inlet
D_{it}	Inner diameter of turbine inlet
D_p	Hydraulic diameter
D_h	Atomizer hydraulic mean diameter
D_{ref}	Reference diameter
d_j	Diameter of dilution holes

FA_{th}	Stoichiometric ratio of fuel to air
FPR	Fan pressure ratio
G_{θ}	Axial flux of swirl momentum
G_{x}	Axial thrust
H_{inner_pass},	The inner passage height
H_{dome}	Dome height
J	Momentum flux ratio
K_{outer_liner}	Thermal conductivity of outer liner
K_{inner_liner}	Thermal conductivity of inner liner
L_{liner},	The distance from the atomize outlet to the centre line of dilution
m_{f}	Fuel flow rate
$m_{f,pn}$	Fuel flow rate in each atomizer
$m_{a,ato}$	Total atomization airflow rate
$m_{a,sw}$	Total mass flow through swirler
$m_{a,innerr\_turn}$	Inner turn section cooling air flow rate
$m_{a,inner\_turn}$	Outer turn section cooling air flow rate
m_{outer_turn}	Outer turn section cooling mass flow rate
$m_{a,dome}$	Dome air flow rate
m_{h}	Mass flow rate through the primary holes
$m_{outer\_pri\_hole}$	Air flow rate through outer primary hole
m_{g}	Gas mass flow rate
m_{j}	Jet mass flow rate
$m_{a,inner\_pn}$	Air flow rate in inner atomizer passage

$N_{inner\_turn\_hole}$	Number of holes in the inner turn section
N_{sw_pri}	Number of primary swirler
$N_{sw\_\sec}$	Number of secondary swirler
$N_{f,opt}$	Optimum number of fuel injectors
N_f	Number of fuel injectors
P_1	Total pressure upstream of the hole
P_{t4}	Outlet total pressure
PF	Outlet temperature pattern factor
p_j	Static pressure downstream of the hole
ΔP_{liner}	Pressure loss across the liner
$\Delta P_{sw\_\sec}$	Pressure drop in the secondary swirler
q_{ref}	Reference dynamic pressure head
$R_{out,\sec\_sw}$	Outer radius of swirler
$R_{inner,\sec\_sw}$	Inner radius of swirler
R_c	Compression ratio
SMD	Sauter mean diameter
T_{t3}	Inlet total temperature
$T_{\max}$	Maximum temperature
T_3	Combustor inlet static temperature
T_{pz}	Primary zone temperature
V_{inner_pass}	The inner annular passage velocity
V_{pass}	Annular passage velocity
V_a	Axial velocity
V_e	Volume occupied for fuel evaporation

$Y_{\max}$	Maximum penetration
α	Hole bleed ratio
$\theta_{inner\_dil\_hole}$	Inner liner dilution jet angle
$\theta_{outer\_dil\_hole}$	Outer liner dilution jet angle
$\theta_{outer\_pri\_hole}$	Outer liner primary air jet angle
$\theta_{inner\_pri\_hole}$	Inner liner primary air jet angle
θ_{sw_vane}	Angle of vanes
η	Combustion efficiency,
ρ_3	Combustor inlet air density
ρ_a	Air density
λ_{eff}	Evaporation constant
τ_{vep}	Evaporation time
σ	Surface tension
μ_f	Fuel dynamic viscosity

1 INTRODUCTION

It is widely accepted that climate change is a very serious environmental concern. Levels of carbon dioxide (CO_2) and other emissions in the global atmosphere have increased substantially since the industrial revolution and now increasing faster than ever before. This has already led to dangerous warming in the Earth's atmosphere and relevant changes around. Emissions legislations are going to be stringent as the years will pass. Hydro carbon fuel cost is also increasing substantially; more over this is non-renewable source of energy.

There is an urgent need of novel technologies in field combustors for reducing emission and fuel consumption. Development of novel combustors needs comprehensive understanding of conventional combustors.

Designing a gas turbine combustor is a challenging process including both analytical methods and rig testing. "Combustor design is an art not a science" is a debatable statement for gas turbine companies even today [Murthy, 1984]. Designing a gas turbine combustor involves a large pool of knowledge of empirical equations, experimental data and other parameters for conceptualising the drawing. Gas turbine combustors involve complex three dimensional flows, heat transfer, mass transfer, radiation, droplet evaporation and chemical kinetics. During the last six decades gas turbine combustor technology has undergone substantial development. Combustor pressures and temperatures are increasing gradually nevertheless combustors still have combustion efficiency close to 100 % [Murthy, 1984]. The high cost of rig testing and increased complexity of CFD simulations minimises their use for conceptualization of feasibility design [Suttaford, 1997]. Preliminary design and conceptualization is extensively undertaken based on empirical co-relations prior to advanced developments.

The design and development of gas turbine combustors is a crucial but uncertain part of an engine development process. At present, the design process relies upon a wealth of experimental data and correlations. The proper use of this information requires experienced combustion engineers and even for them the design process is very time consuming. Some major engine manufacturers have addressed the above problem by developing computer programs based on above test and empirical data to assist combustor designers, but all such programs are proprietary. There is a need of developing design methodologies for combustors. The established design methodologies are a substantial contribution to knowledge in field of combustors. The developed design methodologies are useful for preliminary design assessment of gas turbine combustor.

There is an urgent need of novel technologies in field combustors for reducing emission and adaptability to alternative fuels. In this study author has also proposed and studied several novel concepts in field combustors. These concepts include “Hydrogen Synthesis by Kerosene Reforming Combustor”, “Dilution Zone-less Combustor” and “Hybrid Diffusers”. Study on novel combustors and diffuser have also led to contribution to knowledge. Study on novel combustor and diffuser has yielded application of two patent applications with several other publications.

1.1 Thesis structure

Chapter 1 addresses environmental concerns due to gas turbine emissions. This chapter also contains information on depletion of fossil fuels, conventional combustors and low emission combustors. The need for change in conventional combustors, project objectives and contribution to knowledge of work in this study is discussed. Chapter 2, 3, 4 and appendices have separate abstracts and conclusions.

Chapter 2 provides an insight of the work which has been done by other researchers in the field of gas turbine combustor design and performance analysis. This includes correlations and recommended data proposed by different researchers [Cheng, 2010; Murthy, 1988; Mohammad and Jeng, 2009; Mellor, 1990; Lefebvre and Ballal, 2010; Murthy, 1984] for preliminary design and performance analysis of gas turbine combustor. Chapter 2 also contains brief design methodologies developed for different combustors (Single Annular, Double Annular Radial and Axial, Reverse Flow and triple annular combustor) in this study. Detailed design methodologies have been included in the Appendix A. Novel vortex controlled diffuser and hybrid diffusers concepts have been studied. These studies have resulted in number of publications. Details about the work on hybrid and vortex controlled diffusers are presented in Appendix B.

A novel combustor without a dilution zone has been proposed and presented in chapter 3. This chapter includes discussion about the working principle of this novel combustor with a preliminary investigation of its feasibility. A patent has been filed on this concept following a detailed patent search which was performed to demonstrate the contribution to knowledge.

Chapter 4 presents preliminary discussion about a novel “hydrogen synthesis from kerosene reformation” combustor concept. This combustor is based on use of hydrogen produced within the combustor to aid the combustion. A preliminary investigation has been presented in this chapter. This chapter also includes discussion on hydrogen fuelled combustors.

This study has yielded several publications and patents; abstracts and list of selected publications from this study are presented in Appendix C.

1.2 Environmental Concerns in Aviation Industry

Discharging greenhouse gases and particulates into the atmosphere has an impact on global climate. The environmental issues due to the growing air traffic have become significant in the past three decades. Environmental concerns and depletion of fossil fuel resources have become the driving force for research and development for decreasing the fuel consumption, emissions and finding a fuel for future aviation. Additionally, emissions of carbon dioxide, water vapour and oxides of nitrogen (as a consequence of fossil fuel combustion) contribute to global warming. One of the options for decreasing emissions and dealing with fuel scarcity is introducing hydrogen as a fuel for air transport [Haglind et al., 2006].

Even though combustor technology is developing gradually, there is a need for new technology and concepts to satisfy the emission norms to be laid by the ICAO. This would also help in dealing with the fuel scarcity which is a big problem, arising for the world now.

Although the percentage of aircraft emissions is not a substantial amount as compared to other counterparts which contribute to emissions. But aviation industry is increasing at a substantially high rate as compared to other counterparts which puts aviation industry under pressure to decrease the emissions. The combustor for the gas turbine engines must address a wide range of goals in addition to those associated with the low emissions. These operability goals like flame stability, altitude relight capability cannot be compromised. Therefore, low emissions design approaches must always be considered in relation to their effects on performance. Many of the design goals can represent conflicting requirements and they differ for various applications.

1.3 Availability of Fossil Fuel Resources

The tremendous growth in global population and per capita energy demand indicate to the high consumption of crude oil leading to its extinction around 2020 and 2035. This in turn will affect the oil prices. One of the studies carried out by United States Geological Survey suggests that the ultimate recoverable world oil resources would reach around 125 million barrels per day in 2025 [Schnieder and McKay, 2001]. So there is a substantial need of drive towards decreasing fuel consumption and finding alternative fuels.

1.4 Conventional Combustors

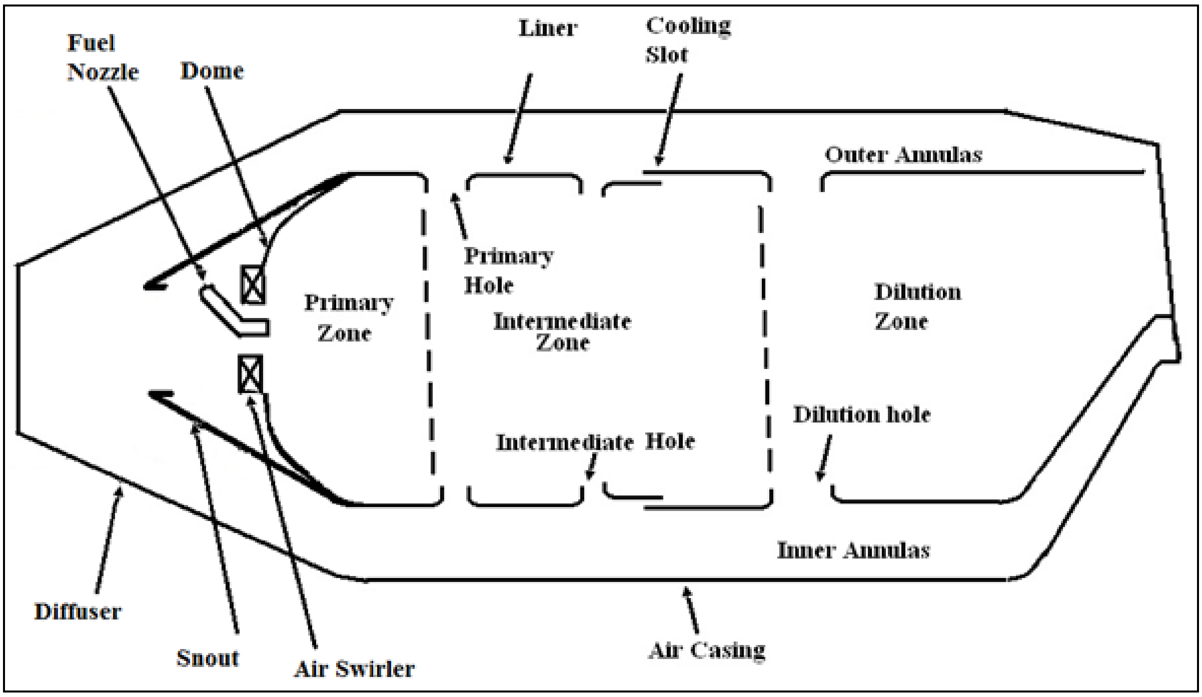


Figure 1-1 Schematic of Conventional Gas Turbine Combustor

Figure 1-1 is a schematic of conventional gas turbine combustor. Location of different components has also been shown. Details of components of combustor have been discussed in section 1.6 of this report.

A gas turbine combustor should satisfy wide range requirements which vary according to the requirements of particular combustor. Basic requirement of almost all combustors is as follows:

- High combustion efficiency.
- Reliable and smooth ignition at ground and at altitude for relights.
- Wide stability limits.
- Low pressure loss.
- No effect of pressure pulsations and other instabilities.
- Low emission
- Size and shape compatibility with a wide range of engines.
- Low cost

- Durability
- Multi fuel capability

In this study low emissions have been considered for further study.

1.5 Types of Combustors

Combustors are divided into three basic types tubular, can-annular and annular. Another type of combustors which is most commonly used is tubo-annular. In can-annular type of combustor many equi-spaced tubular liners are placed surrounded with an annular casing. Configurations of these basic combustors have been shown in Figure 1-2.

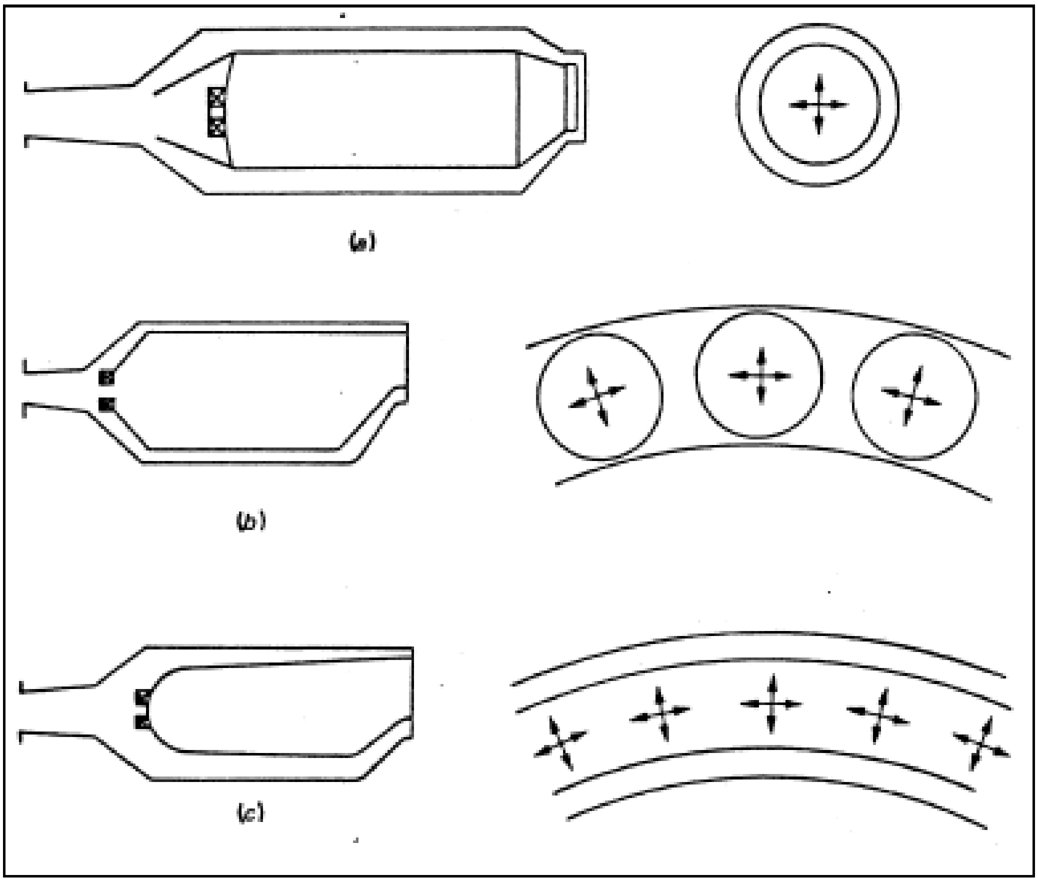


Figure 1-2 Types of Straight Combustors (a) Tubular (b) Can-annular (c) Annular [Lefebvre and Ballal, 2010]

1.5.1 Can or tubular combustion chambers

These are the earliest type of combustors employed in gas turbine engines. These were generally coupled with the centrifugal compressor unit. The compressed air leaving from the centrifugal compressor is separately fed in to tubes which are spaced around the central shaft connecting the turbine and the compressor. Figure 1-3 shows a general layout of such a combustor.

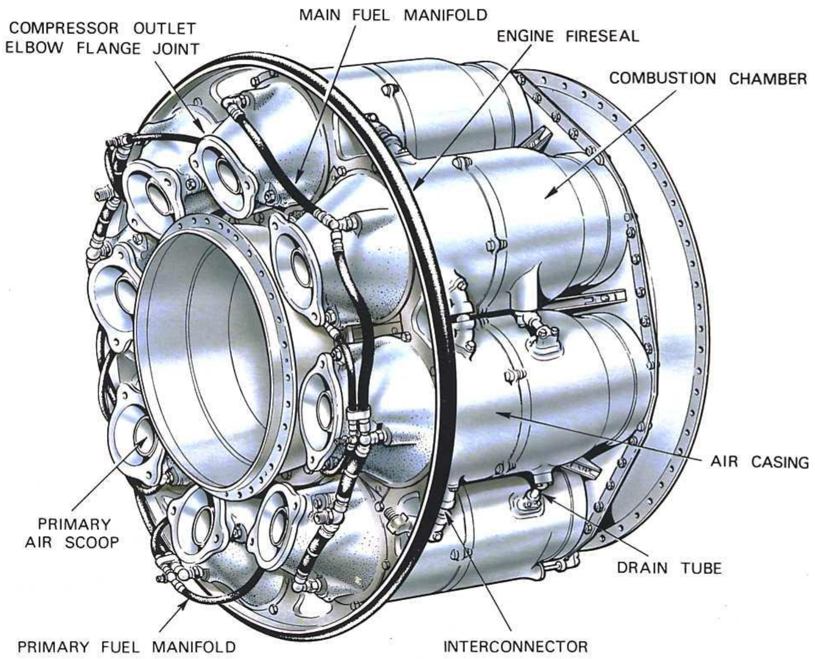


Figure 1-3 Can type combustor [Rolls-Royce, 2005]

1.5.2 Can annular combustor

This type of combustor is an intermediate design between the can type and the annular type combustors. In this type of combustors, many flame tubes are spaced around the center shaft all enclosed in a common air casing. Both tubular and tubo-annular combustors are lengthy and heavy offering less combustion volume for the available combustor placement volume. The figure shows a general arrangement of a can annular combustor in a gas turbine engine

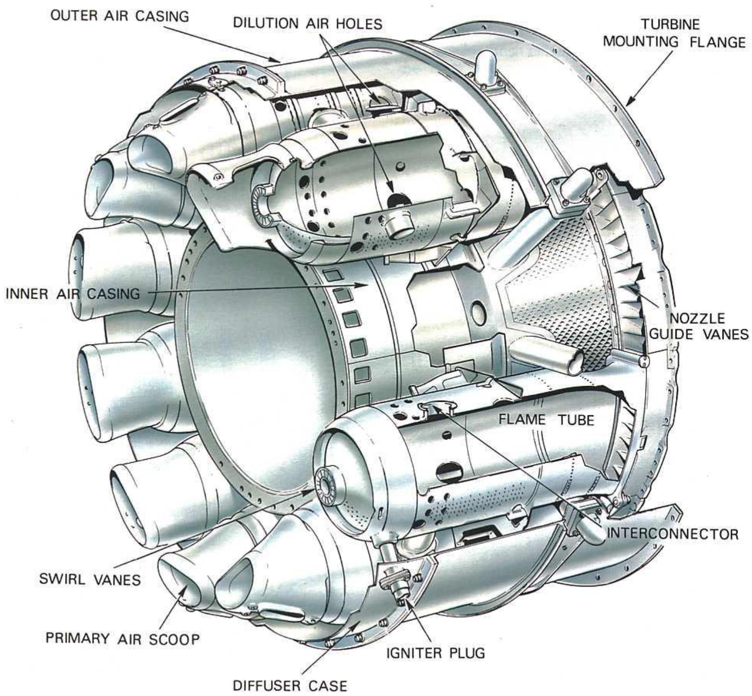


Figure 1-4 Can annular combustor [Rolls-Royce, 2005]

1.5.3 Annular combustors

These are the most evolved of the three types. The combustor consists of an annular flame tube contained between the inner and the outer casings. The inner casing surrounds the center shaft connecting the turbine and the compressor. This design has some major advantage over the other types. Since there are no separate flame tubes around the center shaft, the length can be reduced to get the same combustion volume as that of the other two. This results in reduced weight. Absence of separate flame tubes also means there is no need of inter-connectors. This design significantly reduces the manufacturing cost and maintenance problems. Figure 1-5 shows the general arrangement of an annular combustion chamber in a gas turbine engine.

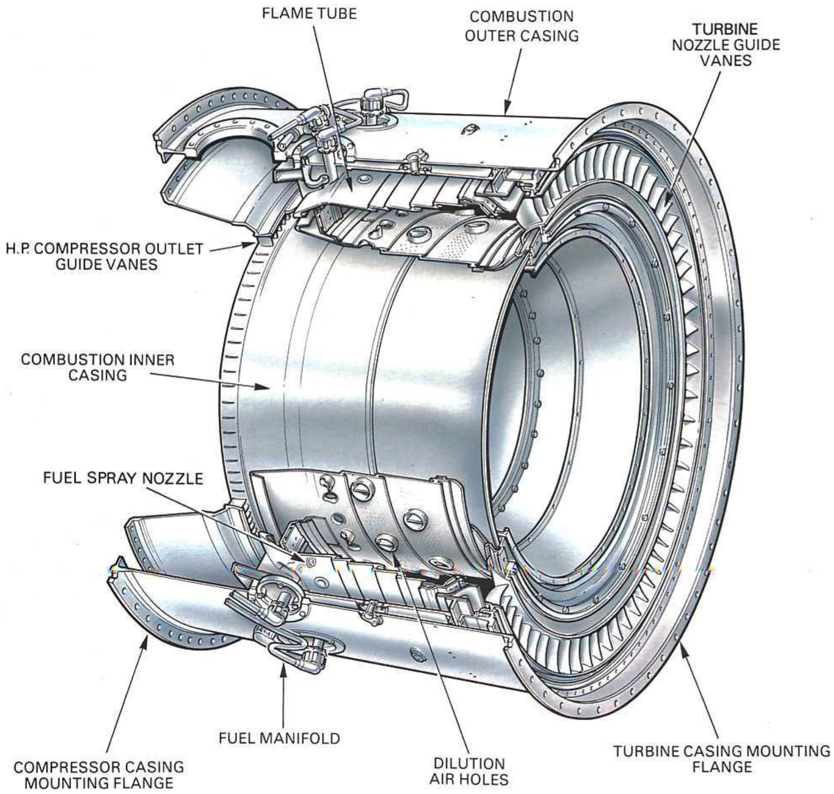


Figure 1-5 Annular combustion chamber [Rolls-Royce, 2005]

1.6 Parts of Combustors

1.6.1 Diffuser

In many aircraft engines compressor outlet velocity reaches velocity of 150 m/s or higher. At such high velocities it is impractical to burn fuel in it; as flame velocity of almost all the fuels is around ~ 5 m/s and if we put fuel at velocity of around 150 m/s, most or some part of fuel would not be burned. This would lead to in-complete combustion, due to which running cost would increase substantially, and lead to pollution. Simplest form of diffuser consists of a diverging passage in which flow is de-accelerated which would eventually lead to rise in static pressure. Figure 1-6 shows the effect of divergence angle on the diffuser. As the divergence angle increases both length and friction losses of diffuser increases, whereas, this leads to increased separation and stall loss. For every area ratio there is an optimal divergence angle at which pressure losses are minimum, this angle is usually between 7° to 12°.

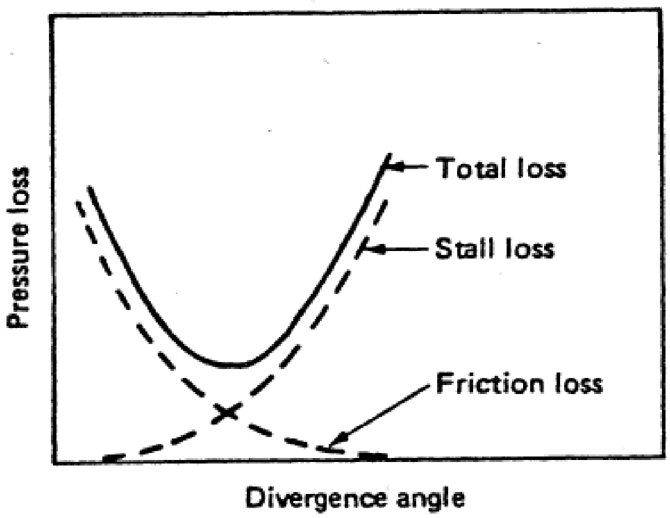


Figure 1-6 Effect of divergence angle on diffuser performance [Lefebvre and Ballal, 2010].

Figure 1-7 shows the schematic of basic faired and dump diffusers. In case of annular (faired) diffusers included angle could be 20-22° [Murthy, 1988]. Conventional diffuser ensures that there is no separation, but it has to pay a penalty in terms of weight, friction and length. To overcome these problems dump diffuser is used, diffusion is achieved partially by conventional diffusion and partially by dumping the in the combustion chamber, ignoring recirculation and sudden expansion.

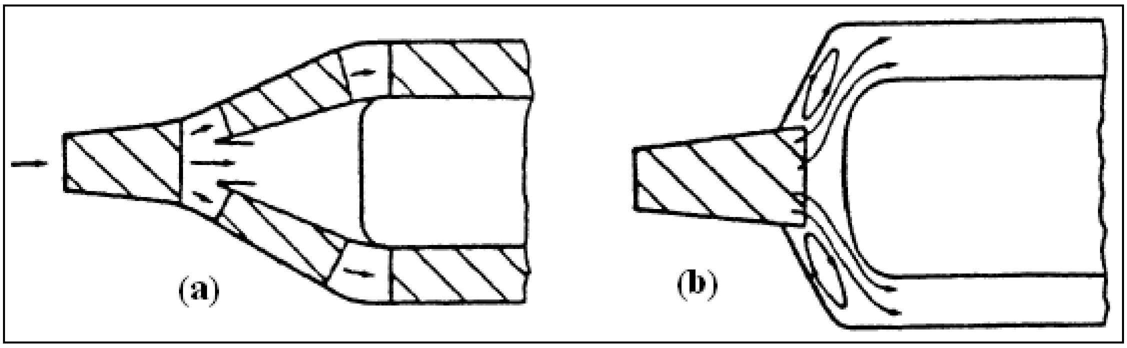


Figure 1-7 Types of diffuser (a) Faired (b) Dump [Murthy, 1988]

A good diffuser is one which has minimum losses with shortest possible length. There are various types of diffuser designs available. The focus of all the different types has been in to minimising the losses and improving the recovery factors. The different types proposed are, faired diffusers, dump diffusers, splitter vanes, vortex controlled diffusers and hybrid diffusers. Figure 1-8 shows a schematic of hybrid diffuser.

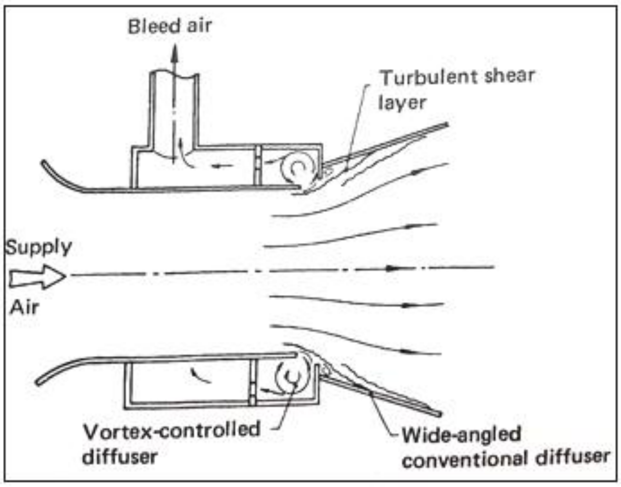


Figure 1-8 Hybrid Diffuser [Lefebvre and Ballal, 2010]

Detailed study on different configurations of hybrid vortex controlled diffusers has been done under this project. In this study effect of vortex chamber, post diffuser angle, bleed angle and other parameters have been studied. All the configurations and results have been presented in Appendix C.

1.6.2 Fuel Injector

The basic need of liquid fuel injector is that liquid fuel must be atomized before being injected in the combustion zone. Fuel injector works on the principle aim of which is to produce high surface to mass ratio in the liquid phase, this results in very high evaporation rates. In most of the injectors atomization is achieved by injecting high velocity liquid to be atomized surrounded by air. Some injectors achieve this by injecting liquid at very high velocity as compared to gas velocity, or an alternate approach to expose low velocity liquid with high velocity air stream. The fuel injectors can be generally classified as twin-fluid atomiser, pressure atomiser and vaporiser.

Figure 1-9, Figure 1-10 and Figure 1-11 represent schematic of pressure atomiser, air-blast atomiser and vaporising system respectively. In vaporising system fuel is heated to vaporise in vaporizing tube and in then injected in the combustion zone. In pressure atomiser and air-blast atomiser, vaporisation is done by help of interaction with air.

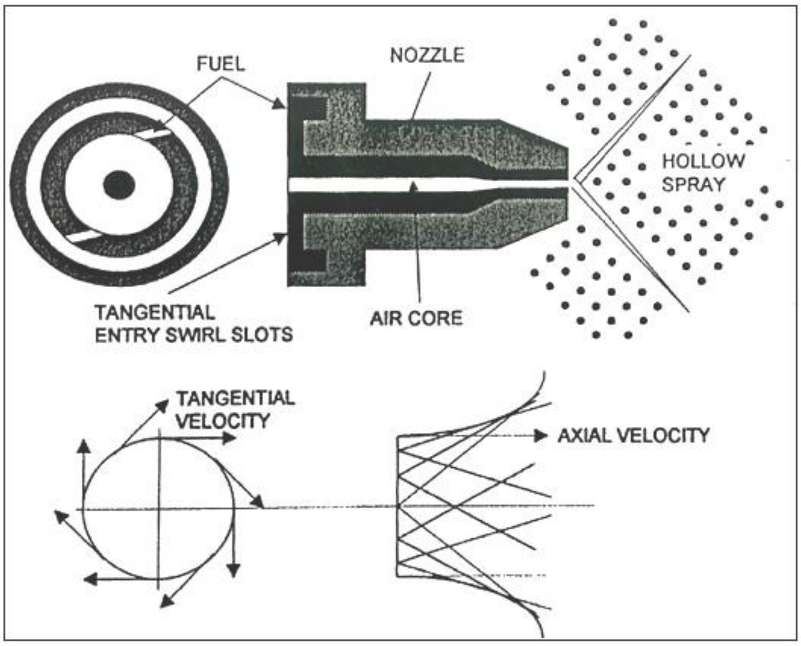


Figure 1-9 Pressure (spray) atomisers [Singh, 2011]

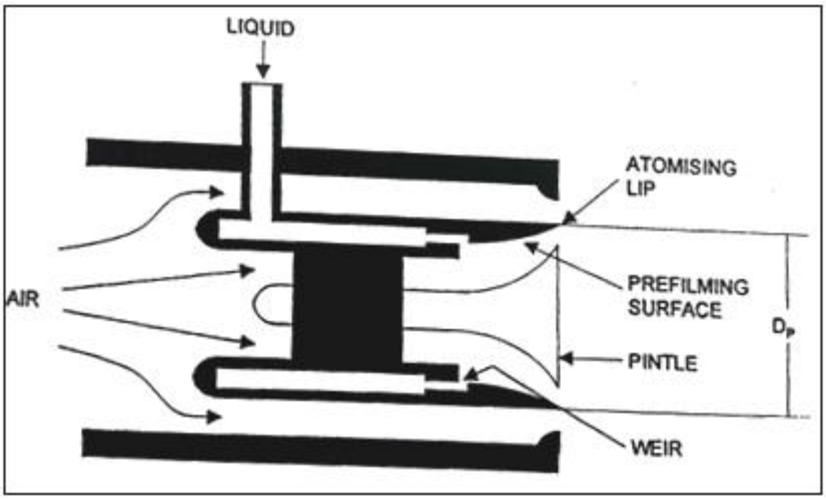


Figure 1-10 Air blast atomisers [Singh, 2011]

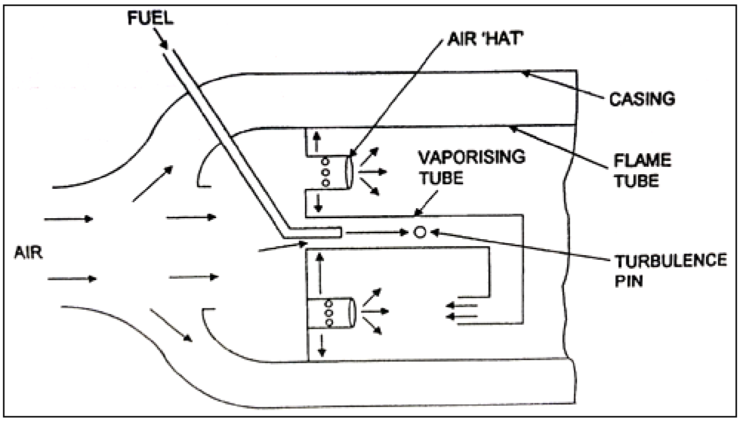


Figure 1-11 Vaporising system [Singh, 2011]

Swirl-spray atomiser produce poorly mixed and atomized fuel air mixture, which leads to areas of both lean and rich equivalence ratios. As the mixture in case of swirl spray atomiser is consisting of both lean and rich mixtures, it shows flame stability limits for higher range of equivalence ratios as compared to other injectors (shown in Figure 1-12). This also leads to increased emissions. On the other hand air blast atomiser atomises and mixed the fuel and air well, which leads to whole mixture being at approximately constant equivalence ratio. Owing to this fact flame stability range in terms of equivalence ratios is shorter as compared to other injectors. Volumetric heat release is highest in case of air blast injectors.

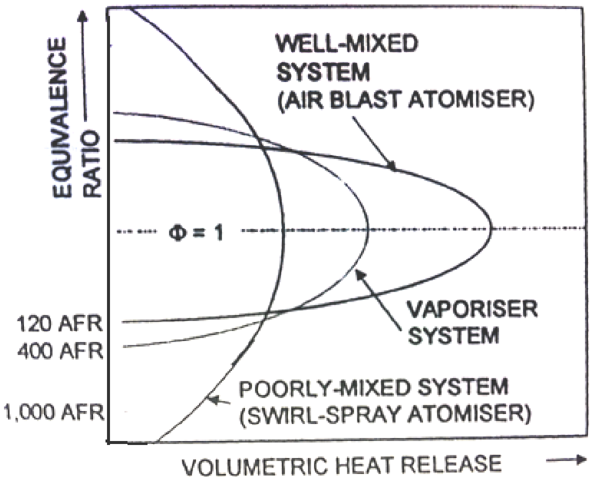


Figure 1-12 Influence of fuel-air mixing on stability limits and volumetric heat release [Singh, 2011]

Fuel properties like viscosity, density, mass transfer number, surface tension plays a vital role in determining the atomizing characteristics of a fuel injector [Murthy, 1988]. The influence of injector and fuel type on combustion efficiency, radiation, can be calculated empirically. Different parameters are calculated for evaluating the performance of fuel injector. One of the important and widely used parameter is Sauter Mean Diameter (SMD), “it is defined as the diameter of a drop having the same volume/surface ratio as the entire spray” [Lefebvre and Ballal, 2010].

1.6.3 Liner

The liner is also called as flame tube. As the name suggests, all the combustion process occurs inside the liner. The liner is split in to three main zones and the air is ingested in to the combustion zone in steps through the holes cut in the liner. The combustor cooling is essentially cooling employed to the liner. There can be many ways of cooling including film cooling, impingement cooling etc.

1.6.4 Air Casing

The air casing is the casing around the combustor. It contains the flame tube, swirler, fuel injectors, fuel plumbing in to the injectors and the ignitors. The gap between the air casing and the flame tube is called passage and it serves the different zones of combustor with appropriate air flow rates.

1.6.5 Combustion Zones

The overall air to fuel ratio in a classic combustor is very high; of the order of 80:1. It is obvious that no fossil fuel can initiate and sustain combustion at such air to fuel ratios. This necessitates the division of combustor in to zones due to which, stable combustion is achieved. Generally, a combustor is split in to three main zones.

Primary zone: It is a zone in to which only a small portion of the air is drawn in (typically 15-20% of the total combustor flow) [Lefebvre and Ballal, 2010]. The air is passed through the swirler and mixed thoroughly with the fully atomized fuel and then ignited. The combustion occurring here is usually rich at full throttle.

Intermediate zone: The combustion occurring in the primary zone may be rich or incomplete at times. And also, the temperature in the primary zone sometimes can reach the stoichiometric flame temperature of the fuel. This necessitates employment of a region in the liner which would draw in just enough air to complete the combustion and partially bring down the temperatures where ideally there is minimum emission. The residence time of the gases in the region is a critical design issue.

Tertiary zone: There are three main objectives of tertiary region. The tertiary region should utilise all the remaining air left even after the complete combustion, bring down the temperature of the effluent gasses to acceptable vales for components situated downstream and while doing so, it must also generate an exit temperature profile to maximise the life of the high pressure turbine.

1.6.6 Swirler

It is a basic requirement or all gas turbine combustors to be able to operate at a wide range of operating conditions which includes injection of rain, ice, low pressure and temperatures. Airflow in primary zone plays a vital role in stabilizing the flame in combustions chamber. Creation of a toroidal flow reversal for recirculation of hot combustion products into the incoming air and fuel is mostly done in all the combustors to stabilise the flame. One of the important ways of producing recirculation is by introducing the swirlers. This kind of recirculation provides better mixing as compared to other methods like bluff bodies. Single and double swirlers are used according the requirement of the combustor. Swirler could be radial also for special cases, but mostly axial swirlers are employed [Lefebvre and Ballal, 2010]. Vanes of swirlers are usually flat, but curved vanes are sometimes preferred for improved aerodynamics.

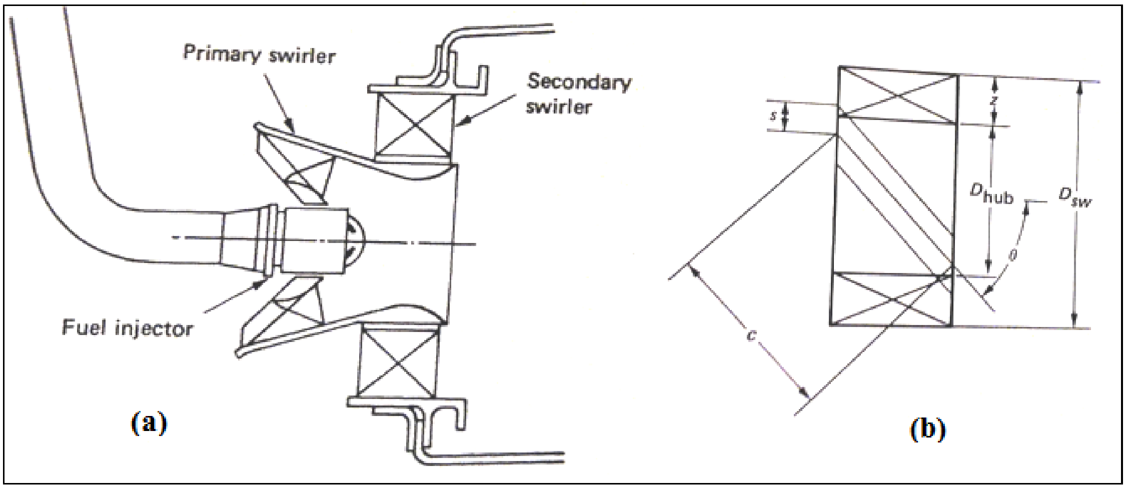


Figure 1-13 Types of swirler (a) double swirler arrangement (b) single flat vane switler; θ = vane outlet angle, c = chord, s = space, z/c = aspect ratio, s/c = space/chord ratio

Swirler performance is calculated by parameter swirl number (S_N). The air flow pattern in primary zone is essential to the combustor performance. The primary zone creates a reversal flow which stabilizes the flame and contributes to the mixing of combustion products with the fresh air and unburnt fuel, hence benefiting rapid and complete combustion in short length [Lefebvre, 1999].

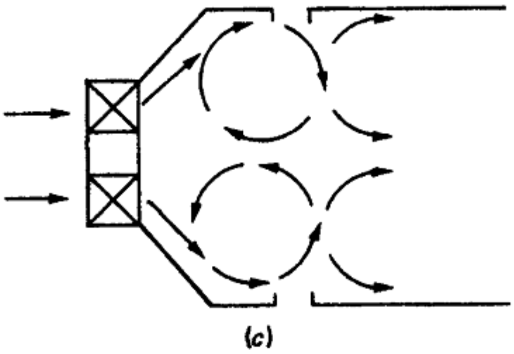


Figure 1-14 Reverse flow produced by swirling flow and opposed jet [Lilley and Gupta, 2004]

Although several ways can be used to produce a reversal flow, a common way in gas turbine combustors is to use swirling flow combined with opposed jets shown in Figure 1-14. The strong swirling flow produces adverse pressure gradients, hence forming a recirculation zone in primary zone [Lilley and Gupta, 2004; Lefebvre and Ballal, 2010]

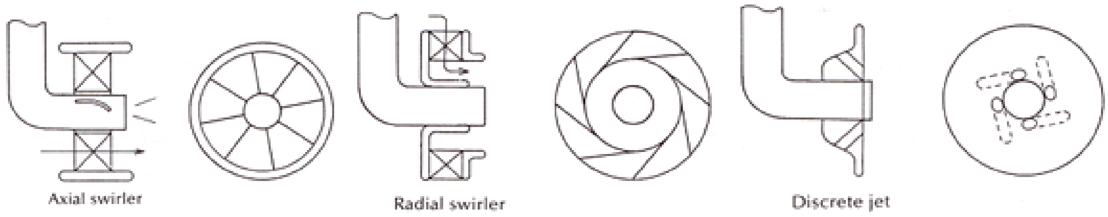


Figure 1-15 Typical swirler types [Lefebvre, 2010]

The swirler can either be designed with single passage where all the flow through the swirler is imparted a unidirectional swirl or with a double passage where, the flow is split in to two paths of counter rotating swirl being imparted to it. Flat and curved vanes are widely used in practical swirlers. Curved vanes produce better performance than flat vanes because they can prevent effectively the flow separation in the vane passages. The swirler with curved vanes can create a larger recirculation zone and induce more air to recirculate. However, the obvious advantages of flat vanes are less cost and easy manufacture. The other advantages include better flame stabilization and lower combustion noise [Lefebvre, 1999].

1.7 Low Emission Combustors

Several designs of new combustors for low emissions have been proposed by different companies and researchers to decrease emissions by gas turbine combustors. Figure 1-16 shows trend of NO_x, CO and UHC with varying equivalence ratio from lean to rich. From Figure 1-16 it is observed that the NOx levels are maximum at air to fuel ratio just below the stoichiometric ratio in the lean regime. With the current demand of combustor temperature rise has increased, the NO_x formation is becoming a thoughtful issue. The process of NO_x is mainly governed by the Zeldovich mechanism [Schwerdt, 2006]. Other important factors in thermal NO_x formation are the residence time, which describes how long the combustion gas has to spend at high temperature.

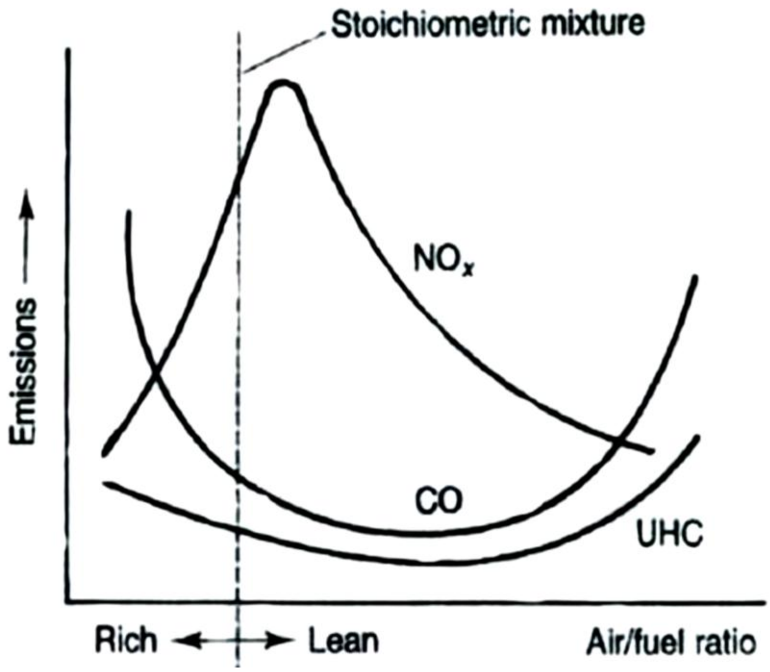


Figure 1-16 Emissions production with varying equivalence ratio [Singh, 2010]

Figure 1-17 shows the range of temperature between which the emissions are in acceptable limits. Of all the emission products from a combustor, NO_x gets special attention due to peculiar behaviour of this substance in air as pollutant. NO_x combines with O_2 to form ozone in the lower atmosphere (troposphere) Ozone in troposphere is injurious to the living beings and hence called bad ozone. NO_x if released in the upper atmosphere causes depletion of the ozone layer in the stratosphere. Ozone in this layer is good ozone and its depletion causes intense UV rays to enter in to earth's atmosphere. Due to these reasons, there are strict regulations imposed on the emission levels all around the world. Presently the emissions regulations are only for landing and take-off cycle, there is no regulation for emissions during the cruise.

There are three different ways by which majority of NO_x is formed in gas turbine combustors.

1. **Fuel bound NO_x**: There is slight amount of nitrogen is present in the fuel; the combustion of such fuel leads to formation of nitrogen oxides which are generally termed ad fuel bound NO_x.

2. **Prompt NO_x**: There are certain conditions in which NO_x emissions are formed especially in low temperature fuel rich flames. The mechanisms involved are not yet fully understood and prompt NO_x levels cannot be predicted with any degree of precision. However, for the conditions that exist in the modern combustor, it is likely that they will be between 0-30 ppm by volume[10], with low values occurring at high temperatures.
3. **Thermal NO_x**: Oxides of nitrogen, of which the predominant one being the nitric oxide are produced by the oxidation of atmospheric nitrogen in high temperature regions of the flame. The process in endothermic and proceeds at significant rated only at temperatures above 1800K. Thus, NO_x arises in the hot central regions of the combustor and their levels are highest at full power conditions.

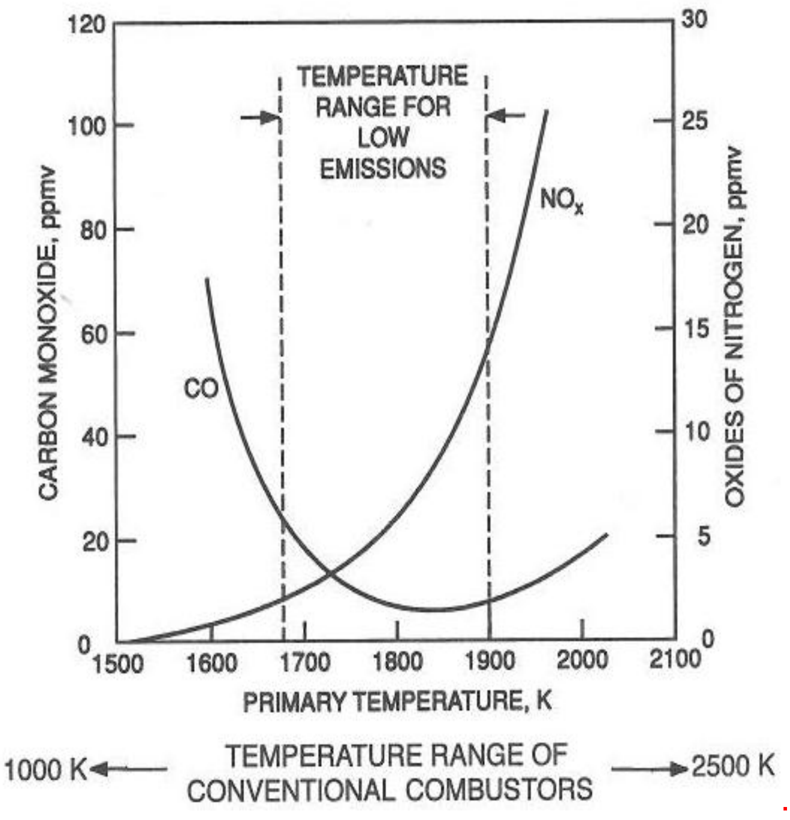


Figure 1-17 Variation of CO and NO_x with primary zone temperature [Lefebvre, 2010]

Form Figure 1-17 it is evident that the range of temperature between which the emissions are acceptable is between 1700K and 1900K, which is very narrow.

To achieve this emission level, the combustor should be designed to have temperatures not above 1900K anywhere inside the flame tube and should still be able to produce the required heat output and temperature profile. To achieve the NO_x emission requirements, there are many methods proposed and they can be classified under either wet or dry method for low NO_x.

There are several methods to reduce NO_x emissions have been proposed by different researchers. Some methods to reduce emissions have been discussed in following subsections.

1.7.1 Wet methods for emission control

The wet methods of emission control deals with liquids injected in to the combustor or upstream of combustor to bring down the peak temperature inside the combustion chamber to air reduction in NO_x levels. The liquid used is usually demineralised water.

1.7.1.1 Water injection in the compressor:

In this method, fine droplets of demineralised water in sprayed at entry or any section through the stages of the compressors. This has two folds effect on the working of the engine. First of all, the water injection decreases the temperature of compressed air thereby increasing the engine cycle efficiency but with decreasing air temperature and water in the flow, the compression needs more energy hence higher torque is demanded from the turbines. The water injection also helps in cooling down the combustion temperature. But since the compressor air is split at the inlet of the flame tube, only a fraction of the water gets in to the primary region; the maximum temperature region of the combustor.

1.7.1.2 Water injection in the diffuser:

In this method, demineralised water is sprayed at the entry or exit of the diffuser, situated just upstream of the flame tube. The advantage of this method compared to water injection in the compressor is that the water would not have absorbed the heat in the compressor hence the cooling capacity is higher. But the disadvantage is that only a fraction of the water injected gets in to the primary zone and the rest flow in to the flame tube thorough intermediate and dilution holes. This decreases the output of the combustor hence compensating measure should be taken in the design of fuel scheduling and air injection in to the combustor.

1.7.1.3 Water injection in the primary zone:

In this method, the water injection is done in the primary zone of the combustor. The main advantages of this method is total volume of water injected is completely in the maximum temperature region and there are no losses. This enables the maximum cooling as water converts in to steam absorbing latent heat of vaporization and then heat is absorbed until temperature equilibrium is attained in the primary zone. The major design concern here is freezing of combustion products. This means, if the combustion products are suddenly cooled down, then certain intermediate species like CO which are reasonably stable get settled and do not oxidise additionally, generating higher level of emissions.

1.7.1.4 Steam injection in the combustor:

The steam injection method is a method in which steam is injected in to the primary zone of the combustor. The major disadvantage of such a system is that they are very hard to realize in a practical propulsion system although there are some examples of them being installed on ground based gas turbines. Another drawback being, the steam is already hot and the cooling capacity of steam is very less compared to water, hence larger volume of steam in needed for same cooling capacity compared to water injection.

1.7.2 Dry Low NO_x Combustors

The other method that can be adopted for designing low emission combustor is not to use any external injection of water or steam but to incorporate certain design changes to the combustor which would result in lower emissions. The changes are in geometry of combustors or in the fuel scheduling of combustor or in the way of utilisation of the available air in the combustor. The various methods by which this is achieved are discussed in subsections below.

1.7.2.1 Variable Geometry Combustors

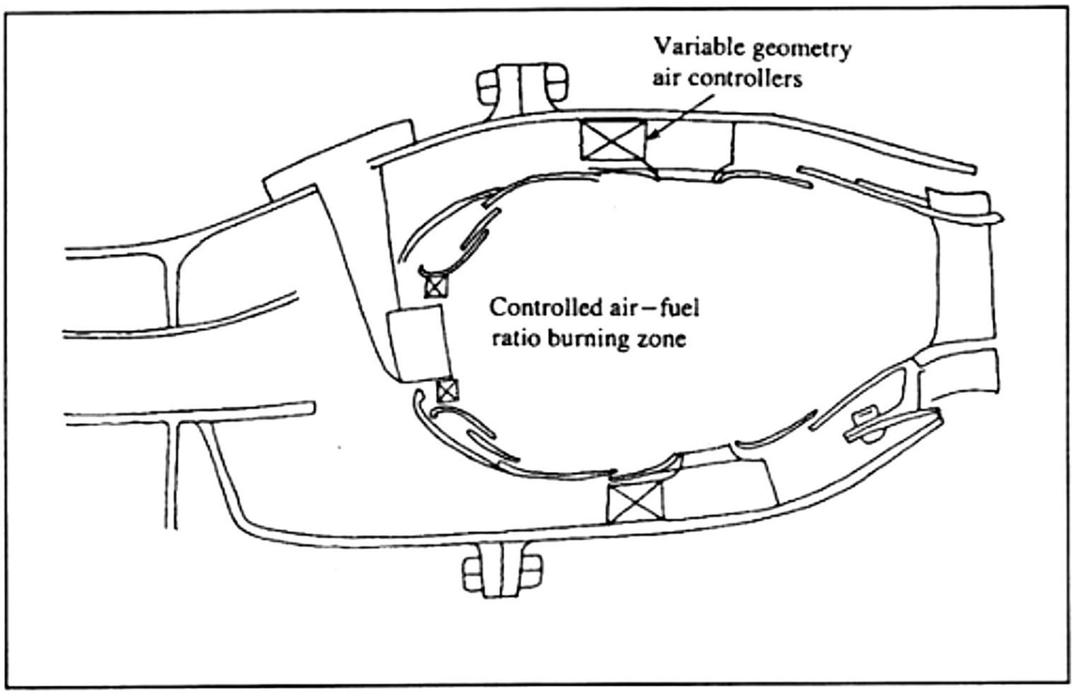


Figure 1-18 Variable geometry combustor [Singh, 2011]

Figure 1-18 shows a schematic of a combustor based on principle of variable geometry. Combustors based on this concept maintain the temperature in primary zone by varying the air to fuel ratio in primary zone. Variation in air supplied to primary zone is achieved by variable geometry. This means maximum volume of air diverted in to primary zone at maximum power and maximum volume of air diverted to the dilution zone at minimum power there by maintaining the primary zone temperature low. One of the major drawback of variable geometry combustor is increased complexity in control and feedback mechanism.

1.7.2.2 Lean Pre-mixed Pre-vaporised combustor

The principle of this type of combustor is to eliminate local regions of high temperature within the flame by supplying the combustion zone with a completely homogeneous mixture of fuel and air. And operate the combustor at equivalence ratio very close to the lean blow out limit. This leads to decrease in NO_x emissions.

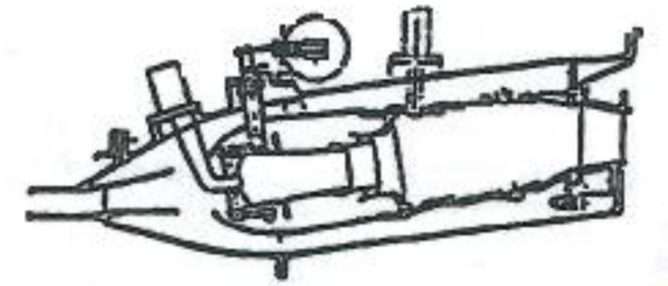


Figure 1-19 LPP Conbustor with variable swirl vanes [Singh, 2011]

One of the main problem of LPP systems is the long period required for fuel vaporising and mixing, this may cause auto ignition before the primary zone of combustor. The chances of flash back problem in a completely vaporised and pre mixed system is considerable.

1.7.2.3 Catalytic Combustor

The principle of catalytic combustor is shown in Figure 1-20. The fuel is injected at the inlet of the combustion chamber and mixed thoroughly with the air. The fuel-air mixture then flows through the catalyst bed or reactor which may consist of several stages each made of different kind of catalysts. After the catalytic bed, a thermal reaction zone is usually provided to raise the temperature of the gases to the required TET and reduce CO and UHC to acceptable values. The major disadvantage of this system is that it is very difficult to design a catalyst that will ignite the fuel air mixture at low compressor exit temperature corresponding to low power settings. The exit temperature requirements of a modern combustor are well above the stability limits of the currently available catalysts.

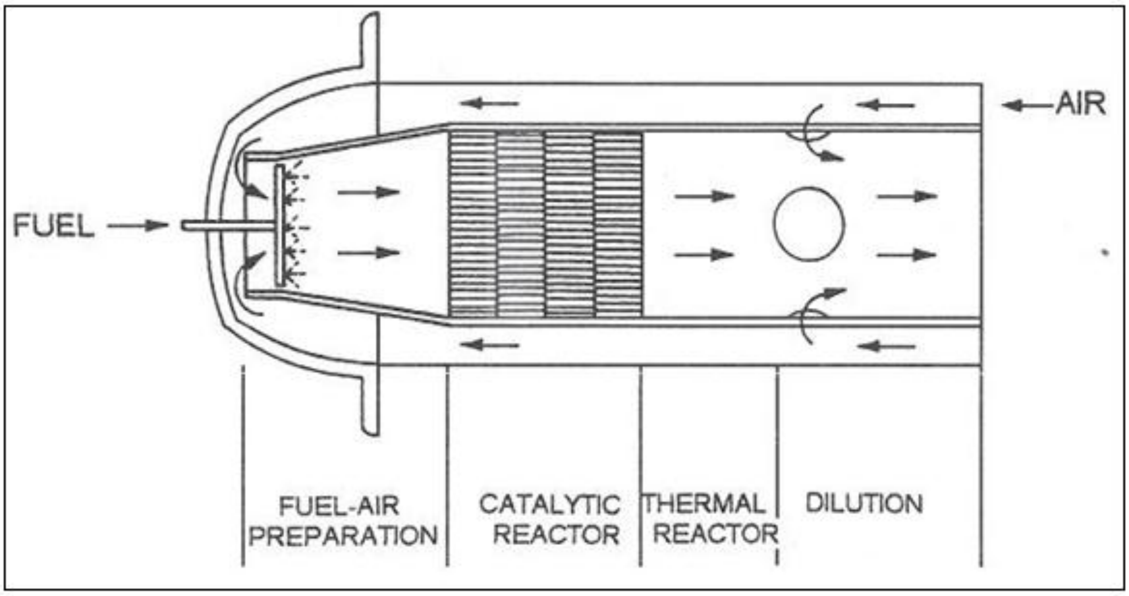


Figure 1-20 Catalytic Combustor [Singh, 2011]

1.7.2.4 Rich-burn Quick-quench Lean-burn (RQL) Combustor

The principle of RQL combustor is to burn rich for low flame temperatures then quickly quenching it and burning lean. Figure 1-21 shows trend of NO_x formation with equivalence ratios. It can be observed that high NOx formation happens at equivalence ratios close to one. In RQL combustor fuel-air mixture is burned rich for keeping the lower flame temperatures. The burning mixture is quickly quenched by help of air addition while keeping an aim to reduce the time of quenching. In case of long time for quenching fuel-air mixture would burn near stoichiometric ratio leading to higher NO_x production. The air-fuel mixture is then burned lean. Time taken during the quenching is a major factor for reduction in NO_x emissions.

The major design challenge for the RQL combustors is the design of quenching section. If there is insufficient air admitted in this section, then the emission would be at the maximum possible limits. There are many companies which are developing this combustor to reduce the time taken for quenching.

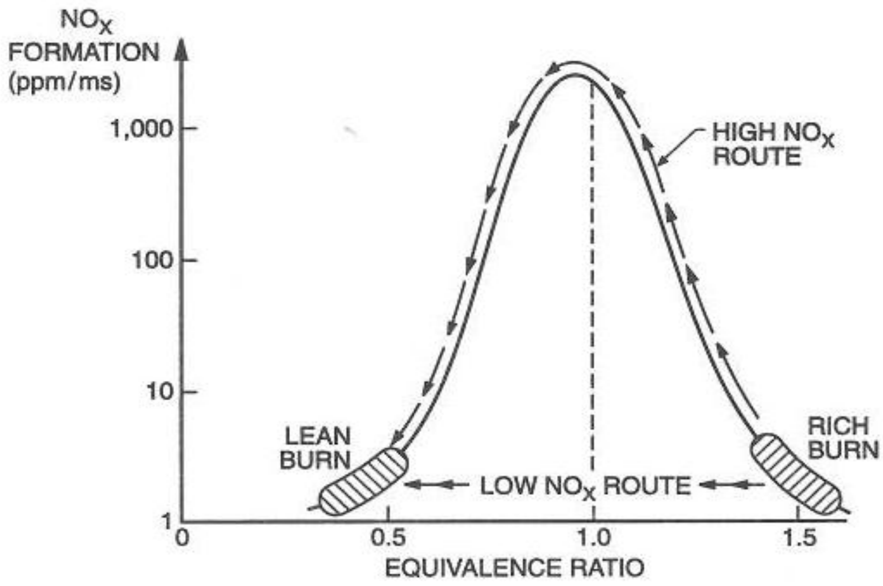


Figure 1-21 Graph to illustrate the principle of RQL combustor [Singh, 2011]

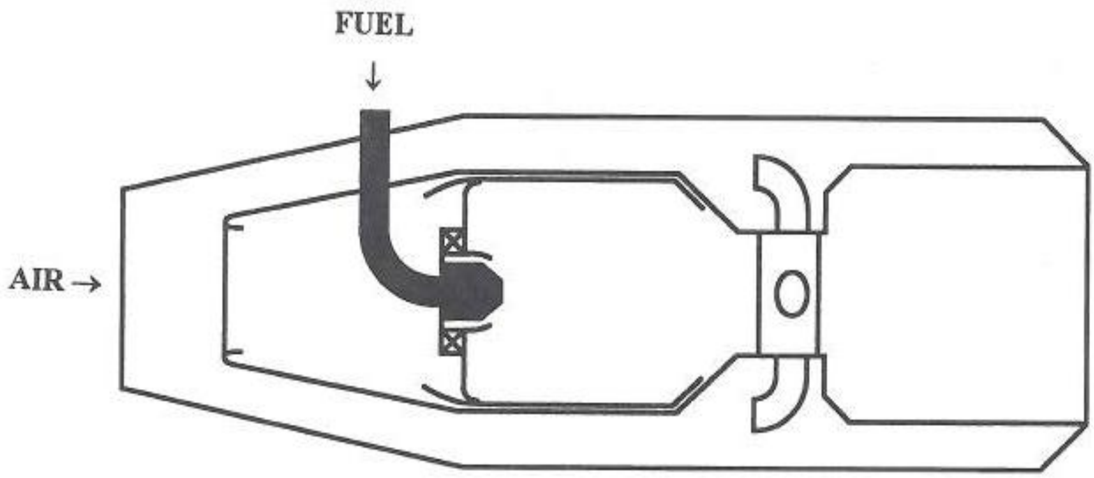


Figure 1-22 RQL Combustor [Singh, 2011]

1.7.2.5 Staged Combustors

Gas temperature distributions in flame tube substantially influence emissions. When gas temperature in primary zone is below a certain value, generally 1670K, the emission index of CO and UHC arises rapidly; while the NOx index will climb fast if gas temperature is higher than 1900 K. One of the most effective ways to reduce emissions generated by a gas turbine engine is to stage the fuel combustion [Breikin, 2006]. Staged combustion is one of the most actively researched techniques proposed to reduce pollutants for aero and industrial gas turbine engines. Staged combustion is also termed dry low emissions (DLE) since there is no need for the injection of water or steam to achieve a significant reduction in pollutant levels. This makes staged combustion more economical and viable for an aero gas turbine engine. Figure 1-23 shows the principle on which staged combustors works.

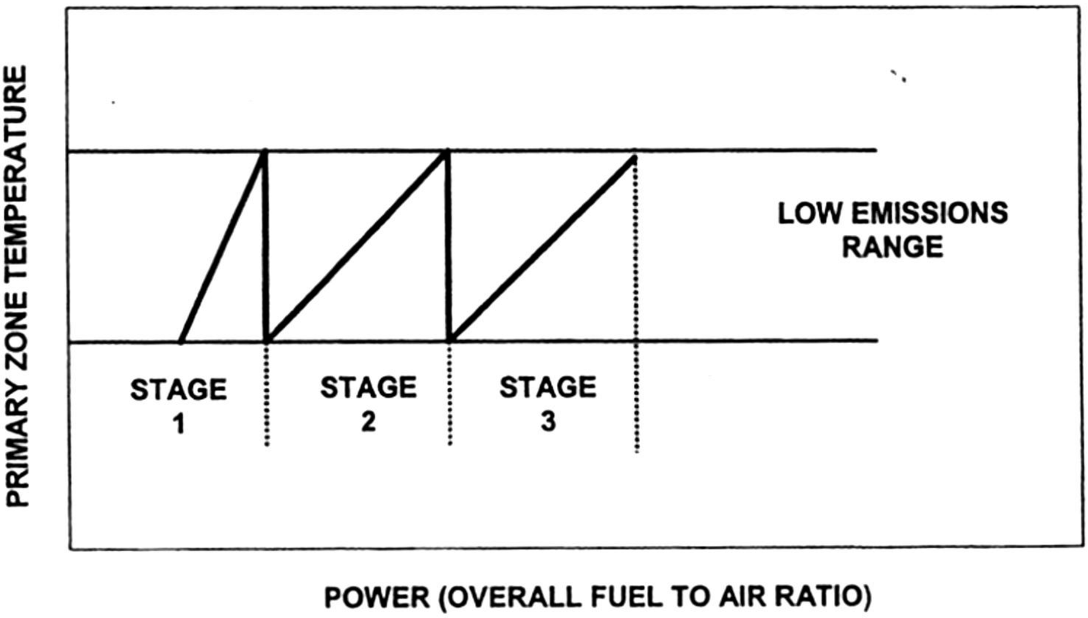


Figure 1-23 Principle of staged combustion [Singh, 2011]

A typical staged combustor has a highly loaded primary zone which provides the entire temperature rise needed to drive the engine at idle and low power conditions. It operates at an equivalence ratio of around 0.8 to achieve high combustion efficiency and low emissions of CO and UHC. At higher power settings, the primary fuel injector acts as a pilot injector and the primary zone as pilot source of heat for the main combustion zone which is supplied with fully premixed fuel air mixture. When operating at maximum power conditions, the equivalence ratio in both zones is kept low at around 0.6 to minimise NO_x and smoke. There are two types of staging one is axial and another is radial. Figure 1-24 shows a GE solution for double annular radially staged combustor. In this combustor main zone and pilot zone both are used on full power, whereas only main staged is used for cruise. This approach can be successfully implemented to achieve lower emissions, lower equivalence ratios, more importantly the design would be roughly of the same length as a conventional combustor.

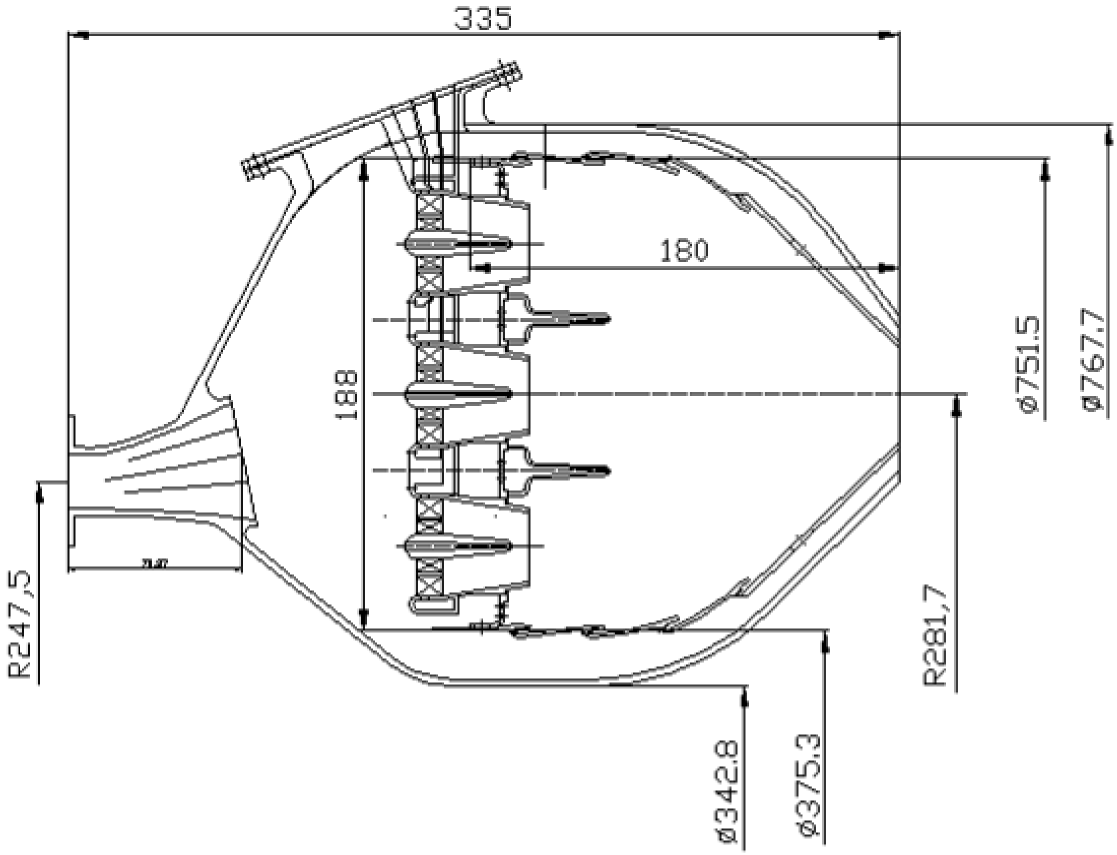


Figure 1-24 Radially staged combustor [Khandelwal, 2012 (f)]

1.8 Objectives and Scope

Gas turbine Combustion is a complex interaction of, fluid dynamics, heat and mass transfer and chemical kinetics. At present, the design process relies upon a wealth of experimental data and correlations. The proper use of this information requires experienced combustion engineers and even for them the design process is very time consuming. Some major engine manufacturers have addressed the above problem by developing computer programs based on above test and empirical data to assist combustor designers, but those programs are not available in public domain.

In the present work, development of design methodologies for different conventional combustors has been undertaken. The methodologies could be used to carry out preliminary design along with prediction of the cooling slots to evaluate heat transfers and temperatures for existing combustion chambers. Such comprehensive study should provide ample opportunity for the designer to make the right decisions at early stage of design process. It should also be an effective study aid.

There is an urgent need of novel technologies in field combustors for reducing emission and fuel consumption. In this study two novel concepts of combustors have been proposed. Preliminary investigation on feasibility of the novel combustors has been done and presented in this thesis. These novel concepts include “Hydrogen Synthesis by Kerosene Reforming Combustor”, “Dilution Zone-less Combustor” and “Hybrid Diffusers”. A novel concept of hybrid diffuser with different configurations has also been studied in this study.

1.9 Contribution to knowledge and technology transfer

Established design methodologies are a substantial contribution to knowledge in the field of combustors. The developed design methodologies are useful for preliminary design assessment of a gas turbine combustor. Study on novel combustors and diffuser have also led to contribution to knowledge. Study on novel “Hydrogen Synthesis by Kerosene Reforming Combustor”, “Dilution Zone-less Combustor” and diffuser has yielded application of two patent applications with several other publications. A detailed list of publication has been included in Appendix C.

Author has also technically helped and worked with a number of MSc researchers for their project which has led to technology transfer.

2 CONVENTIONAL COMBUSTORS - DESIGN

2.1 Abstract

The design and development of gas turbine combustors is a crucial but uncertain part of an engine development process. Combustion within a gas turbine is a complex interaction of, fluid dynamics, heat and mass transfer and chemical kinetics. At present, the design process relies upon a wealth of experimental data and correlations. The proper use of this information requires experienced combustion engineers and even for them the design process is very time consuming. Some major engine manufacturers have addressed the above problem by developing computer programs based on above test and empirical data to assist combustor designers, but all such programs are proprietary. There is a need of developing design methodologies for combustors. The established design methodologies are a substantial contribution to knowledge in field of combustors. The developed design methodologies are useful for preliminary design assessment of gas turbine combustor.

In the present work, developments of step by step design methodologies for various combustors have been done. These methodologies assist in preliminary design and evaluation of conventional, advanced and novel gas turbine combustion chamber. Methodologies developed could be used to carry out preliminary design along with prediction of preliminary performance of the designed combustor. The combustors for which design methodologies have been developed are single annular combustor, double annular radial and axial combustor, triple annular radial and axial combustor and reverse flow combustor. Combustor design methodology includes detailed designing of different parts of combustors which includes diffuser, fuel injector, swirler, casing, liner, cooling holes and other relevant parts.

Developing design methodologies has resulted in several publication, abstracts are available in Appendix C.

2.2 Introduction

Designing a gas turbine combustor is a challenging process including both analytical methods and rig testing. “Combustor design is an art not a science” is an debatable statement for gas turbine companies even today [Murthy, 1984]. Designing a gas turbine combustor involves a large pool of knowledge of empirical equations, experimental data and other parameters for conceptualising the drawing. Gas turbine combustors involve complex three dimensional flows, heat transfer, mass transfer, radiation, droplet evaporation and chemical kinetics. During the last six decades gas turbine combustor technology has undergone substantial development. Combustor pressures and temperatures are increasing gradually nevertheless combustors still have combustion efficiency close to 100 % [Murthy, 1984]. The high cost of rig testing and increased complexity of CFD simulations minimises their use for conceptualization of feasibility design [Suttaford, 1997]. Preliminary design and conceptualization is done as much as possible based on empirical co-relations prior to advanced developments.

Discharging greenhouse gases and particulates into the atmosphere has an impact on global climate. Environmental concerns and depletion of fossil fuel resources have become the driving force for research and development for decreasing the fuel consumption, emissions and finding a fuel for future aviation. Additionally, emissions of carbon dioxide, water vapour and oxides of nitrogen (as a consequence of fossil fuel combustion) contribute to global warming. One of the options for decreasing emissions and dealing with fuel scarcity is introducing hydrogen as a fuel for air transport [Haglind, 2006].

Even though combustor technology is developing gradually, there is a need for new technology and concepts to satisfy the emission norms to be laid by ICAO. Developing combustor technology would also help in dealing with the depleting fossil fuel resources.

Although the percentage of aircraft emissions is not a substantial amount as compared to other counterparts which contribute to emissions, the aviation industry is increasing at a substantially high rate as compared to other counterparts which puts aviation industry under pressure to decrease the emissions. Green lobby argues that air travel is a luxury not a necessity.

Subsections in this chapter present the work which has been done in this study; which includes development of step by step design methodologies for various combustors. These methodologies assist in preliminary design and evaluation of conventional. Methodologies developed could be used to carry out preliminary design along with prediction of the cooling slots for a given metal temperature limit or to evaluate heat transfers and temperatures for existing combustion chambers. The combustors for which design methodologies have been developed are single annular combustor, double annular radial and axial combustor, triple annular radial and axial combustor and reverse flow combustor. Combustor design methodology includes detailed designing of different parts of combustors which includes diffuser, fuel injector, swirler, casing, liner, cooling holes and other relevant parts. It is to be noted that design methodology for single annular combustor have been taken from a source in public domain which have been verified and adapted according to requirement in the current study. Prediction of emissions have been done by correlations provided by various researchers, detailed analysis of emissions have not been included in this study as other students in Department of Power and Propulsion are working on it. The design methodologies would be used to develop a code for combustor design, which would be eventually used in a big project for combustor design names DEPTH.

Section 2.3 describes general preliminary combustor design methodology. This includes preliminary design of combustor parts proposed by different researchers [Cheng, 2010; Murthy, 1988; Mohammad and Jeng, 2009; Mellor, 1990; Lefebvre and Ballal, 2010; Murthy, 1984]. Section 2.4 briefly describes various design methodologies developed for designing various combustors. Detailed methodologies have been explained in Appendix A.

There is an urgent need of novel technologies in field combustors for reducing emission and adaptability to alternative fuels. In this study author has also proposed and studied several novel concepts in field combustors. Study on novel combustors and diffuser have also led to contribution to knowledge. Study on novel combustor and diffuser has yielded application of two patent applications with several other publications.

2.3 General Combustor Design Methodology

Over the last few decades, large amount of experimental data has been collected from research carried out in industry and universities across the world has enabled one to develop a theoretical design methodology for design of a gas turbine combustor. The general procedure for a conventional combustor design is shown in a step by step process in Figure 2-1.

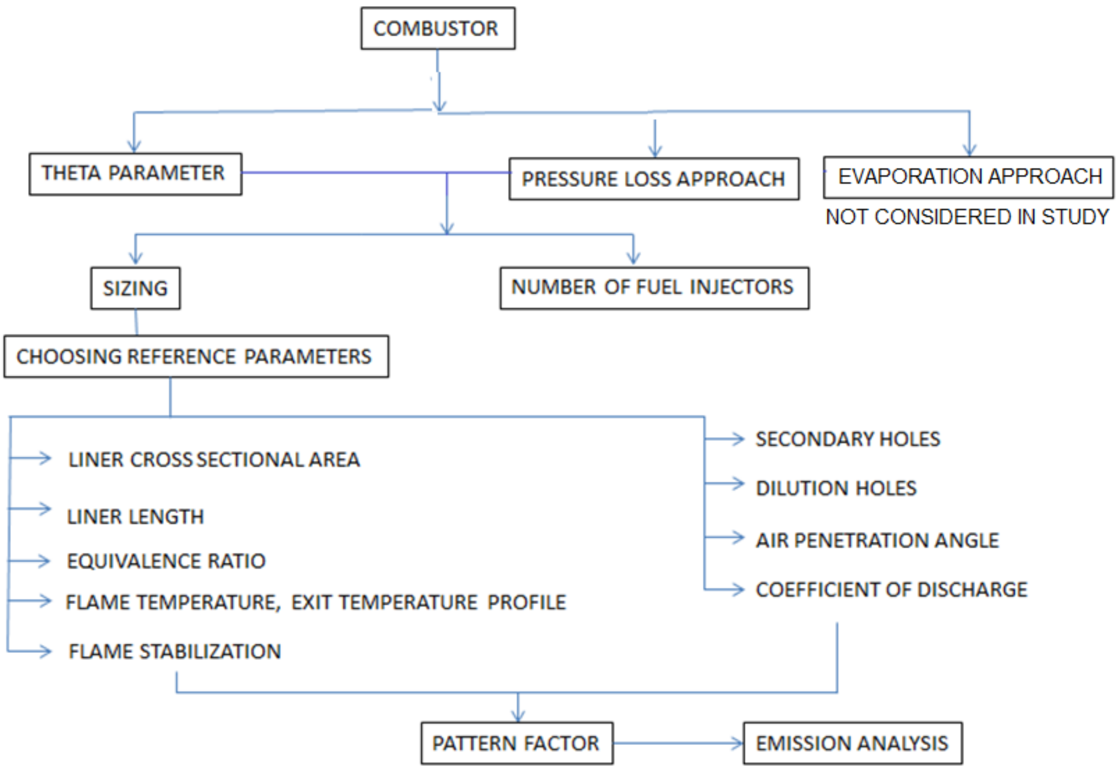


Figure 2-1 Flow chart of classic combustor design methodology.

In general, gas turbine combustors are sized based on their application. That is, to design a combustor for flying applications, the most important criteria is reliable altitude relight characteristics whereas to design a ground based gas turbine combustor, minimum pressure loss is most desired.

For preliminary combustor sizing of flying applications of gas turbine, theta parameter approach [Mohammad, 2009] is used and for ground applications, pressure loss approach is used.

Theta parameter approach: The combustor is sized based on a parameter called theta. The theta parameter can be mathematically written as,

$$\boldsymbol{\theta} = \mathbf{P_3^{1.75} A_{ref} D_{ref}}^{\mathbf{0.75}} \left[\frac{\mathbf{exp}\left(\frac{\mathbf{T_3}}{\mathbf{300}}\right)}{\mathbf{W_3}} \right] = \boldsymbol{\eta_c} \qquad \text{Equation 2-1}$$

For any particular combustion chamber, a few measurements of combustion efficiency at various air mass flow rates or inlet pressures is sufficient to define the form of η_c Vs θ curve. It is then possible to read off values of η_c for any specified valued of pressure temperature and mass flow for any flight conditions. The pressure and mass flow are a function of altitude and thus combustion efficiency will decrease with decreasing P_3. This means the combustion efficiency is least at the maximum ceiling of the engine. Hence in the theta parameter approach, the maximum operating altitude and a minimum acceptable value of combustion efficiency at this altitude are set to determine the minimum casing area. Thus after some iterations, the combustion efficiency nearer to the set value could be obtained for an acceptable area of cross section.

The particular advantage of theta parameter approach is that it permits drastic reductions in number of low pressure combustion efficiency measurements that have to be made on an engine during development. The Figure 2-2 shows the relation between combustion efficiency and theta parameter for existing engines. If the value of the theta parameter obtained from calculations is greater than 1.1×10^{10} then the combustion efficiency of the combustor is said to be about 99.99%.

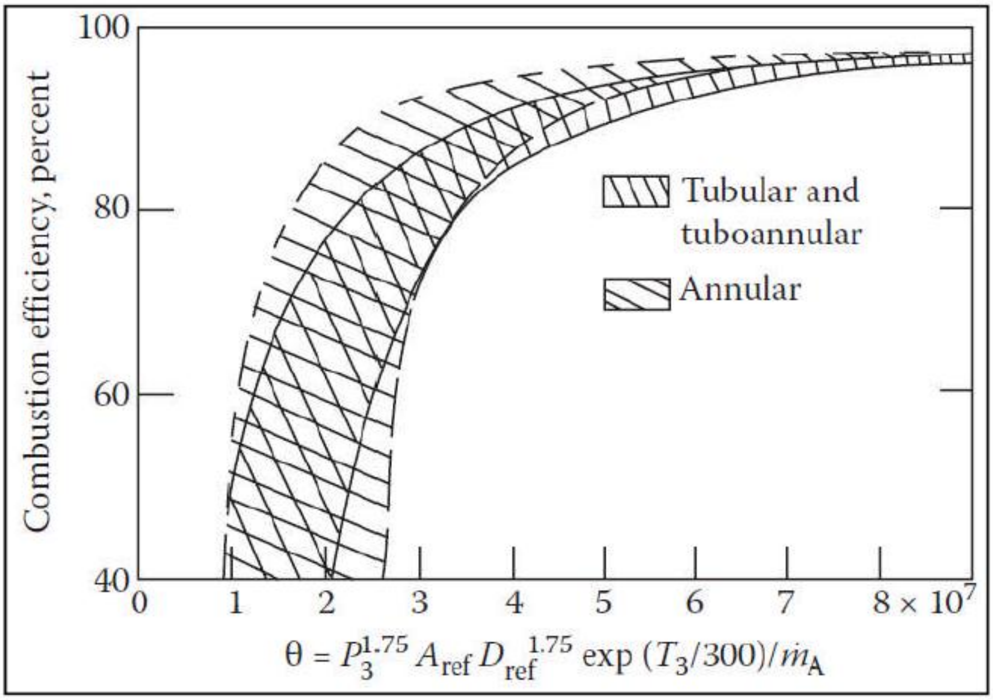


Figure 2-2 Relation between combustion efficiency and theta parameter [Lefebvre, 2011]

Pressure loss approach [Murthy, 1988]:

The pressure loss for incompressible flow in a duct can be written as,

$$\frac{\Delta P}{P} = \frac{\Delta P}{D} \times \frac{R}{2}\left[\frac{M\sqrt{T}}{A_c P_2}\right]^2 \quad \text{Equation 2-2}$$

Or in combustor nomenclature, this can be expressed as,

$$\mathbf{A_{ref}} = \sqrt{\frac{\mathbf{R}}{\mathbf{2}} * \left(\frac{\mathbf{W_3}\sqrt{\mathbf{T_3}}}{\mathbf{P_3}}\right)^2 \times \left(\frac{\Delta\mathbf{P}/\mathbf{q_{ref}}}{\Delta\mathbf{P}/\mathbf{P_3}}\right)} \quad \text{Equation 2-3}$$

Where $\Delta P/P_3$ is termed as overall pressure loss. It does not include the combustion losses (hot losses). The term $\Delta P/q_{ref}$ is called as pressure loss factor. This term is very important for the design engineer as this is total resistance introduced to the air stream between the compressor and the combustor outlet. This is a fixed property of the combustion chamber and has a typical value of 25 [Singh, 2011]. The equation 2.3 relates the flow parameters, the pressure loss factor and the temperature parameters at the inlet of the combustor to the reference area of the combustion chamber. Hence for a ground based gas turbine combustor, having insignificant variation in inlet pressure and temperature, the relation yields a minimum reference area of the combustor.

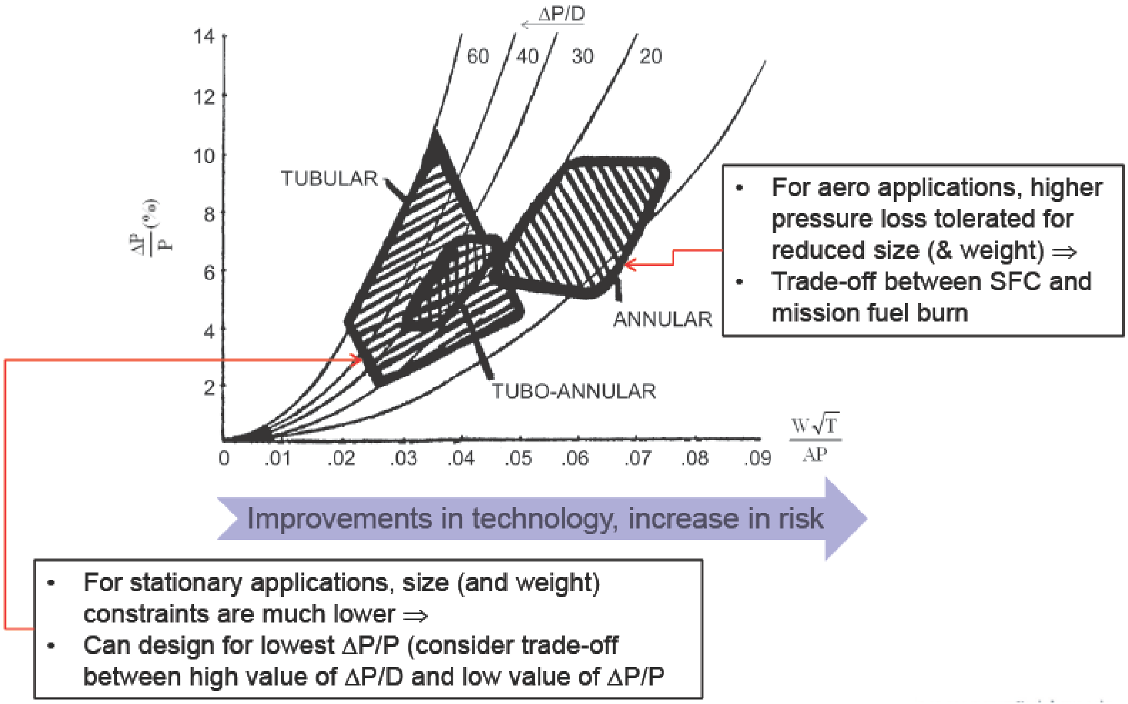


Figure 2-3 Pressure Loss Approach: Design Chart [Singh, 2011]

The two approaches discussed above lead to basic sizing of the combustor. In case of jet engines, combustor sizing is done using both methods and the larger of the two is selected. This is done to maintain maximum reliability during altitude relight.

The diffuser design is an important step in complete sizing of the combustor since the reference velocity and the flow quality needed for further calculations are the output of the diffuser. Fishenden and Stevens [1977] studied the performance of annular combustor dump diffuser. They proposed that the optimum area ratio for a conventional part of dump diffuser is around 1.8 and rest part is dumped. This is joined at 90° to the reference area. They have said that factors determining the losses in dump and settling length are a) amount of diffusion b) radius of curvature of flow which depends on size and shape of flame tube and dump gap. It is also proposed that diffuser to dome length is selected based on the criterion:

$$\frac{Diffuser\ to\ Dome\ Length}{Prediffuser\ exit\ height} \geq 1.2$$

Preliminary diffuser design can be done by following some simple mathematical steps as discussed below.

A). choose the area ratio AR (ratio of combustor reference area to the diffuser reference area)

B). Calculate the length to width ratio (LW):

C). Calculate the ideal pressure recovery coefficient (C_{pi}):

D). Calculate the actual pressure recovery coefficient (C_p):

E). Calculate the pressure loss in the aerodynamic diffuser as a fraction of its inlet total pressure:

F). Calculate the pressure loss in the dump diffuser as a fraction of its total inlet pressure:

G). Calculate total pressure loss in the diffuser section:

H). Check θ and it should be within 4° to 8°:

I). Check the length to width ratio LW. It should be in the prescribed range. (Consider weight etc.)

J). Plot ε_t vs. AR and choose the value of AR that corresponds to the minimum ε_t.

The next step in the design of the combustor is to utilise the available air completely in an efficient way without compromising the combustion stability and the heat release rate whilst maintaining least possible emission level. To do so, only a portion of the air is admitted in to primary zone. This zone is sized for maximum combustion efficiency at idling throttle setting. This inevitably leads to very low combustion efficiency at full throttle and altitude conditions. Hence, the intermediate zone is then sized for maximum combustion efficiency at full throttle and at altitude conditions. The remaining air flow is then progressively admitted in to the combustor through dilution holes. Careful design of dilution zone results in maintaining required exit temperature profile and exit maximum temperature. Once the combustor is divided in to three zones, and the heat release rate found, it is very important to utilize the available air in each zone for cooling of liner as well. This then leads to detailed design of the combustor.

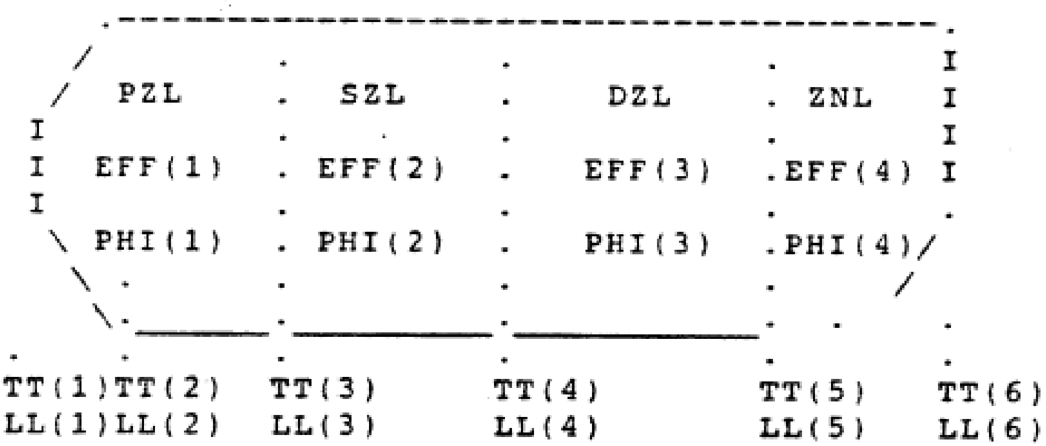


Figure 2-4 Liner in its modular form [Murthy, 1988].

TT = Total Temperature; LL = Zonal Length

L/D ratio of each zone is decided based on experience and recommended data. L/D ratio selected is represented in Table 2.1

	P	S	D	N
Ground	0.8	0.4	1.4	0.6
Airborne > 10 km	0.5	1.0	0.7	0.6
Airborne ≤ 10 km	0.55	0.85	0.9	0.4

Table 2-1 Relative Zonal Length [Murthy, 1988]

Minimum combustor volume is calculated based on empirical equation proposed by Herbert [1962] which is shown below;

For Φ ≤ 1.0

$$\boldsymbol{V_{min} = 6.403E-5\ \frac{m_{pz}\,T_{pz}^{1.5}\,e^{\left(\frac{21200}{T_{pz}}\right)}\eta\,(1+\frac{\phi\eta}{ARFS})^2}{P_2^2(1-\eta)(1-\phi\eta)}} \qquad \text{Equation 2-4}$$

For Φ > 1.0

$$\boldsymbol{V_{min} = 6.403E-5\ \frac{m_{pz}\,T_{pz}^{1.5}\,e^{\left(\frac{21200}{T_{pz}}\right)}\eta\,(1+\frac{\phi}{ARFS})^2}{P_2^2(1-\eta)(\phi-\eta)}} \qquad \text{Equation 2-5}$$

For selecting number of cans it is suggested that prime numbers like 7, 11, 13, 17 and so on are selected to avoid any possible resonance due to multiple frequencies. Number of cans is calculated by following correlation.

$$\dot{m}_{can} = \frac{1.0\, e^{-5} \times D_{ft}^{2} \times P_3 \times (100.-OPL)}{\sqrt{T_{pz}}} \quad \text{Equation 2-6}$$

$$Number\ of\ Cans = \frac{\dot{m_{pz}}}{\dot{m_{can}}} \quad \text{Equation 2-7}$$

Murthy [1988] used following correlations for designing swirlers due to lack of empirical correlations relations, following correlation proposed by Northern Research Engineering [Northern Research Engineering, 1974] is used. However the range of Dft does not cover values of annular combustor.

$$D_{sw} = Outer\ Dia\ = 0.225\, D_{ft}\ + 0.01905\, m \quad \text{Equation 2-8}$$

$$D_{hub} = Inner\ Dia\ = 0.100\, D_{ft}\ + 6.35 \times 10^{-3}\, m \quad \text{Equation 2-9}$$

Following relation is used by Murthy [1988] to calculate swirl number

$$S_N = \frac{2}{3}\,\frac{1-(D_{hub}/D_{sw})^3}{1-(D_{hub}/D_{sw})^2} \tan\theta \quad \text{Equation 2-10}$$

$\theta =$ Vane angle

If swirl number is greater than 0.6 it is considered as strong swirl, otherwise weak swirl. Vane angle is generally kept between 45-60 degrees. Usually number of blade is between 8-16.

Mohammad [2009] proposed following methodology for designing axial swirler. Two axial swirlers are to be designed. Each one has a hub diameter (*hs D*) and a tip diameter (*t s D*). The primary swirler will provide the air necessary for the atomization process. It will have a hub diameter equals to the secondary nozzle swirler chamber diameter (assumed 3 times the outer diameter). The secondary swirler will provide the secondary swirling air. The hub diameter of the secondary swirler will be the tip diameter of the primary swirler (assuming zero thickness).

Vane angle (θ_v) is assumed between 30-60°. Vane thickness (V_t) is assumed in range of 0.7-1.5 mm [Mellor, 1990]. Number of vanes (n_v) is assumed between 8-16 [Mellor, 1990] and the solidity (σ_v) of vanes is assumed around 1.0. Area of swirler (A_{sw}) is estimated according to relation mentioned below:

$$\boldsymbol{A_{sw} = 1/Cos(\theta_v)^* \left(\left(2^* \rho_3^* \frac{pdl}{(k_{sw^*}(\dot{m}_{sw})^2)}\right) + \frac{1}{A_t^2}\right)^{-0.5}} \qquad \text{Equation 2-11}$$

A_l is assumed to be equal to dome area and liner pressure drop (pdl) is equal to swirler pressure drop. K_{sw} is swirling constant 1.3 for flat vanes. Tip diameter is calculated by the following relation.

$$\boldsymbol{A_{sw} = \frac{\pi}{4} * \left(D_{hs}^2 - D_{hs}^2\right) - 0.5 * n_v * v_t * (D_{ts} - D_{hs})} \qquad \text{Equation 2-12}$$

Estimation of swirl number is done as per done by J. N. Murthy [Murthy, 1988]. Vane spacing is calculated by

$$\boldsymbol{v_s = \pi * \frac{(D_{hs} + D_{ts})}{2 * n_v}} \qquad \text{Equation 2-13}$$

$$\boldsymbol{v_c = \sigma_v * v_s} \qquad \text{Equation 2-14}$$

Dilution hole and film cooling calculation by Mohammad and Jeng [Mohammad and Jeng, 2009] have been included in this report.

2.4 Performance Parameters

2.4.1 Fuel Injector Performance

The influence of injector and fuel type on combustion efficiency, radiation, can be calculated empirically. Different parameters are calculated for evaluating the performance of fuel injector. One of the important and widely used parameter is Sauter Mean Diameter (SMD), “it is defined as the diameter of a drop having the same volume/surface ratio as the entire spray” [Lefebvre, 1999]; i.e.

$$\boldsymbol{SMD = \frac{\sum n\, D^3}{\sum n\, D^2}} \qquad \text{Equation 2-15}$$

Mass mean diameter (MMD), is also used frequently to evaluate the atomisation quality. It is the diameter above or below which life 50 % of mass of the drops. “the uniformity of the circumferential distribution of fuel in a conical spray is generally refereed as its patternation” [Murthy, 1988]. Patternation of 80% is mostly acceptable for most applications.

Different empirical co-relations have been proposed by different researchers for determination of SMD for different types of fuel injectors. One of the empirical relation proposed by Murthy [1988] is:

$$\boldsymbol{SMD = K_1\, \sigma^a\, \mu^b\, (\Delta P)^c m_L^d\, \rho_a^e \left(1 + \frac{m_L}{m_a}\right)^f + K_2}$$ Equation 2-16

σ = Surface Tension

K1, K2, a, b, c, d, e, and f are the coefficients to be experimentally determined for each type of injector.

For a typical pressure atomizer [Murthy, 1988]:

a = 0.16 to 0.19 ;b = 0.16 to 0.3 ;c = -0.275 to -0.5

d = 0.25 ;e = -0.1 to -0.25

f and K_2 are applicable for air blast atomiser.

Nukiyama and Tanasawa [Nukiyama and Tanasawa, 1939] conducted a study on air blast atomization. The drop size data was correlated by the following empirical equation for SMD.

$$\boldsymbol{SMD = \frac{0.585}{U_R}\left(\frac{\sigma}{\rho_L}\right)^{0.5} + 53\left(\frac{\mu_L^2}{\sigma\rho_L}\right)^{0.225}\left(\frac{Q_L}{Q_A}\right)^{1.5}}$$ Equation 2-17

Recent work by Rizkalla and Lefebvre [1975], Rizk and Lefebvre [1980], El-Shanawany and Lefebvre [1980] and Rizk [1977] has confirmed the validity of this form of equation for the SMD.

Air blast atomizer is similar to the configuration which has been studied by EI Shanawany and Lefebvre [1980]. According to the experimental and analytical results, EI Shanawany and Lefebvre [1980] gave the following equation to predict the SMD for prefilming air blast atomizers:

$$\frac{SMD}{D_h} = (1+\frac{m_f}{m_{a,ato}})\times[0.33\times(\frac{\sigma_f}{\rho_a \times V_a^2 \times D_p})^{0.6} \times (\frac{\rho_f}{\rho_a})^{0.1} + 0.068\times(\frac{\mu_f^2}{\sigma\times\rho_f \times D_p})^{0.5}]$$

Equation 2-18

Cheng Bo [Cheng, 2010] have used following relations for calculating the SMD which he has represented as D_f.

$$\boldsymbol{D_F = 2.25\left(\frac{\sigma\mu_F}{\rho_3}\right)^{0.25} \Delta P_F^{-0.5}}\ (\boldsymbol{Pressure\ Atomiser})$$

Equation 2-19

$$\boldsymbol{D_F = 10^{-3}\left[1+\frac{1}{AFR}\right]^{0.5}\left[\frac{(\sigma\rho_F)^{0.5}}{\rho_A U_A} + 0.06\left(\frac{\mu_F^2}{\sigma\rho_A}\right)^{0.425}\right]}\ (\boldsymbol{Airblast\ Atomiser})$$

Equation 2-20

2.4.2 Swirler Performance

Beer and Chigier [1972] have proposed an empirical expression for calculation of swirl number for swirlers of constant vane angle.

$$\boldsymbol{S_N = \frac{2}{3}\,\frac{1-(D_{hub}/D_{sw})^3}{1-(D_{hub}/D_{sw})^2}\tan\theta}$$

Equation 2-21

$\theta =$ Vane angle

Pressure losses could be considerable in case of swirlers. Knight and Walker [1957] have proposed empirical correlation for calculation of pressure loss across swirler for swirlers with thin vanes.

$$\dot{m}_{sw} = \left\{\frac{2\rho_3\ \Delta P_{sw}}{K_{sw}\left[(\sec\frac{\theta}{A_{sw}})^2 - \frac{1}{A_L^2}\right]}\right\}^{0.5}$$

$\Delta P_{sw} =$ Total Pressure drop due to swirler;

$A_{sw} = F$rontal area of swirler

$\theta =$ Vane angle;

K_{sw} = 1.3 for flat vane, 1.15 for curved vane

2.4.3 Combustion Efficiency

The combustion efficiency, η, is defined as the ratio of heat released in combustion to the heat supplied. Based on burning velocity model, Lefebvre combined the effects of combustor operating pressure and temperature, and combustor characteristic dimensions to evaluate combustion efficiency. This is the famous "θ" parameter [Rizkalla and Lefebvre, 1975; Rizk and Lefebvre, 1980; Jones, 1978].

$$\eta = f(\theta) = f\left(\frac{P_3 \times A_{ref} \times D_{ref}^{0.75} \times \exp(\frac{T_3}{300})}{m_3}\right)$$

Equation 2-22

$$\theta = \frac{P_3^{1.75} \times A_{ref} \times D_{ref}^{0.75} \times \exp(\frac{T_3}{300})}{m_3}$$

Equation 2-23

Mohammad and Jeng [2009] used following method proposed by Lefebvre [1999]. Lefebvre [1999] proposed following relation for calculation of combustion efficiency.

$$\boldsymbol{\eta_c = f(airflow\ rate)\left(\frac{1}{evaporation\ rate} + \frac{1}{mixing\ rate} + \frac{1}{reaction\ rate}\right)^{-1}}$$

Equation 2-24

$$\boldsymbol{\eta_c = \eta_e \times \eta_\theta}$$

Equation 2-25

2.4.4 Pattern Factor

The parameter which is important for designing a nozzle guide vane is overall temperature distribution factor, which highlights maximum temperature. It is defined as

Pattern Factor $= \frac{T_{max} - T_4}{T_4 - T_3}$

Equation 2-26

Where T_{max} = maximum recorded temperature

T_3 = main inlet temperature

T_4 = mean exit temperature

The temperatures of importance for turbine blades are related to average radial temperature (radial temperature distribution factor). It is measured in terms of profile factor.

Profile Factor = $\frac{T_{mr}-T_4}{T_4-T_3}$ Equation 2-27

T_{mr} = maximum circumferential mean temperature

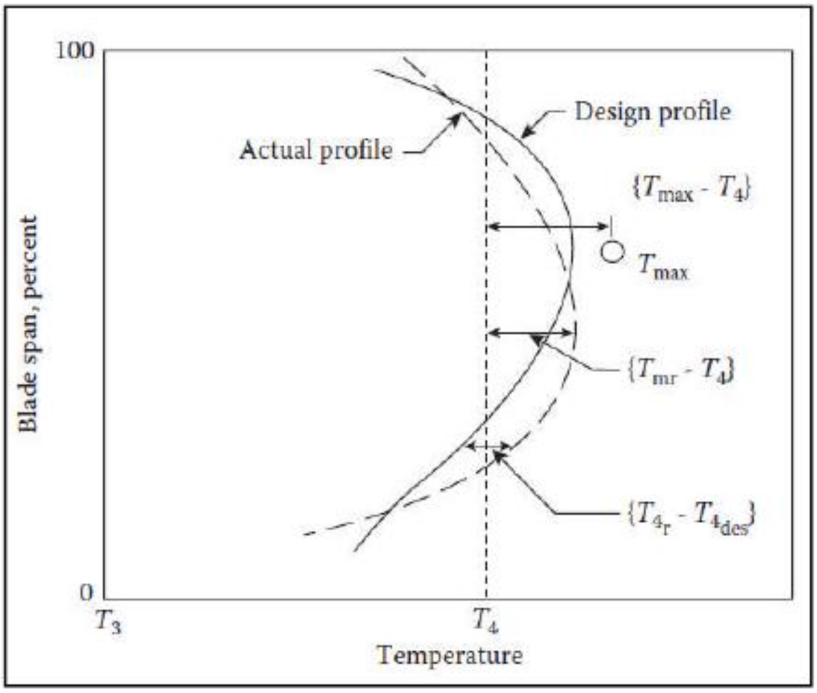


Figure 2-5 Turbine Profile factor [Lefebvre, 2010].

The quality of combustor outlet temperature distribution can be measured by the pattern factor which is essential to the life of turbine components. Two factors have significant effects on the pattern factor. One is the total liner length; another is the pressure loss across the liner. The former is related to the length and time used for dilution; the latter correlates to jet penetration of dilution air. Based on a large amount experiment results, Lefebvre [1999] correlated the pattern factor as:

$$\frac{T_{max}-T_4}{T_3-T_4} = f\left(\frac{L_{liner}}{H_{dome}} \times \frac{\Delta P_{liner}}{q_{ref}}\right)$$ Equation 2-28

For annular combustors, the following equation can be used to predict the pattern factor [Lefebvre, 1989; Gupta et al., 1984].

$$\frac{T_{\max} - T_4}{T_3 - T_4} = 1 - \exp(-0.05 \times \frac{L_{liner}}{H_{dome}} \times \frac{\Delta P_{liner}}{q_{ref}})^{-1}$$ Equation 2-29

Lefebvre [2010] proposed that the pattern factor reduces with the increase of $\frac{L_{liner}}{H_{dome}} \times \frac{\Delta P_{liner}}{q_{ref}}$, but the variation of pattern factor is not evident when $\frac{L_{liner}}{H_{dome}} \times \frac{\Delta P_{liner}}{q_{ref}}$ is over 70.

2.4.5 NO_x prediction

Different NOx production methods have been explained in chapter 1. Based on analyses of experiment results on different conventional aero engine combustors, Lefebvre derived the following correlation to predict the NOx [Lefebvre, 1981].

$$EI_{NOx} = \frac{9 \times 10^{-8} \times (\frac{P_{3t}}{1000})^{1.25} \times V_c \times \exp(0.01 \times T_{st})}{m_{a,3} \times T_{pz}}$$ Equation 2-30

Odgers and Kretschmer [1985] also suggested an equation:

$$\mathrm{NOx} = 29 \times \exp\left(-\frac{21670}{T_C}\right) \times P_3^{0.66} \times \left[1 - \exp\left(-250\tau\right)\right] g / kgfuel$$ Equation 2-31

Lewis [1991] supplied another correlation:

$$\mathrm{NOx} = 3.32 \times 10^{-6} \exp\left(0.008 T_C\right) P_3^{0.5}\, ppmv$$ Equation 2-32

Another equation was summarized by Rokke et al. [1993]:

$$\mathrm{NOx} = 18.1 \mathrm{P}_3^{1.42} m_3^{0.3} q_{\mathrm{c}}^{0.72}\, ppmv$$ Equation 2-33

Meanwhile, Rizk and Mongia [1994] gave their correlation as follows:

$$\mathrm{NOx} = 15.10^{14} \left(t_c - 0.5 t_e\right)^{0.5} \exp\left(-\frac{71100}{T_{st}}\right) P_3^{-0.05} \left(\frac{\Delta P_{3-4}}{P_3}\right)^{-0.5} g / kgfuel$$ Equation 2-34

Appropriate method has to be selected for precise calculation of NO_x emissions for all cases.

2.4.6 CO prediction

Correlations for CO emissions calculation have been developed by Lefebvre [1981], and Rizk and Mongia [1994]. In their correlations, the relevant temperature is not the local peak value adjacent to the evaporating fuel drops, but the average value throughout the primary zone, T_{pz}, which is lower than the peak local temperature due to application of cooling system. Also, because CO emissions are most important at low pressure conditions, where evaporation rates are relatively slow, it is necessary to reduce the combustion volume, V_{cb}, by the volume occupied in fuel evaporation, V_e. Hence, Lefebvre [1981] gave his equation:

$$CO = 86 \mathrm{m}_3 T_{pz} \frac{\exp\left(-0.00345 T_{pz}\right)}{V_c - V_e} \left(\frac{\Delta P_{3-4}}{P_3}\right) P_3^{1.5} g / kgfuel \qquad \text{Equation 2-35}$$

$$V_e = 0.55 m_{pz} D_0^{\ 2} / \rho_{pz} \lambda_{eff} \qquad \text{Equation 2-36}$$

where V_e is proportional to the square of the initial mean drop size. This equation highlights the importance of good atomization to the attainment of low CO emissions. However, this correlation is suitable for liquid fuel combustors.

Rizk and Mongia [1994] suggested another equation:

$$CO = 0.18 x 10^9 \exp\left(\frac{7800}{T_{pz}}\right) / P_3^{\ 2} \left(t - 0.4 t_e\right)\left(\Delta P_{3-4} / P_3\right)^{0.5} \qquad \text{Equation 2-37}$$

This equation yields a slightly lower dependence on combustion temperature and s little higher dependence on pressure than Lefebvre’s approach.

These equations are more likely to be used to predict CO emissions of liquid fuel.

2.4.7 UHC Prediction

Similar to carbon monoxide emissions Lefebvre [1999] and Le Dilosquer [1998] has drawn following correlation for calculation of unburned hydrocarbon.

$$UHC = \frac{11764 \dot{m}_A T_{pz} P_3^{1.25} V_C \exp(-0.00345 T_{pz})}{(V_C - V_E)\left(\frac{\Delta P_L}{P_3}\right) P_3^{2.5}}$$

Equation 2-38

$$UHC = \frac{1810 \dot{m}_A T_{pz} P_3^{1.25} V_C \exp(-0.00345 T_{pz})}{(V_C - V_E)\left(\frac{\Delta P_L}{P_3}\right) P_3^{2.5}}$$

Equation 2-39

Rizk and Mongia [1994] presented the following correlation to predict UHC formation:

$$EI_{UHC} = \frac{0.755 \times 10^{11} \times \exp(9756 / T_{pz})}{P_3^{2.3} \times (\tau - \tau_{evp})^{0.1} \times (\Delta P / P)^{0.6} \times (\Delta P / P_3)^{0.6}}$$

Equation 2-40

Where:

$$\tau_{vep} = \frac{SMD^2}{\lambda_{eff}}$$

Equation 2-41

2.5 Design Methodology for Combustors

Brief description of design methodologies have been presented in subsections of this chapter which included an algorithm. Design methodologies have been developed under technical guidance of the author and collaborative effort of MSc researchers [Yan, 2010; Zhang, 2010]. Detailed design methodologies have been presented in Appendix A.

2.5.1 Single Annular Combustor

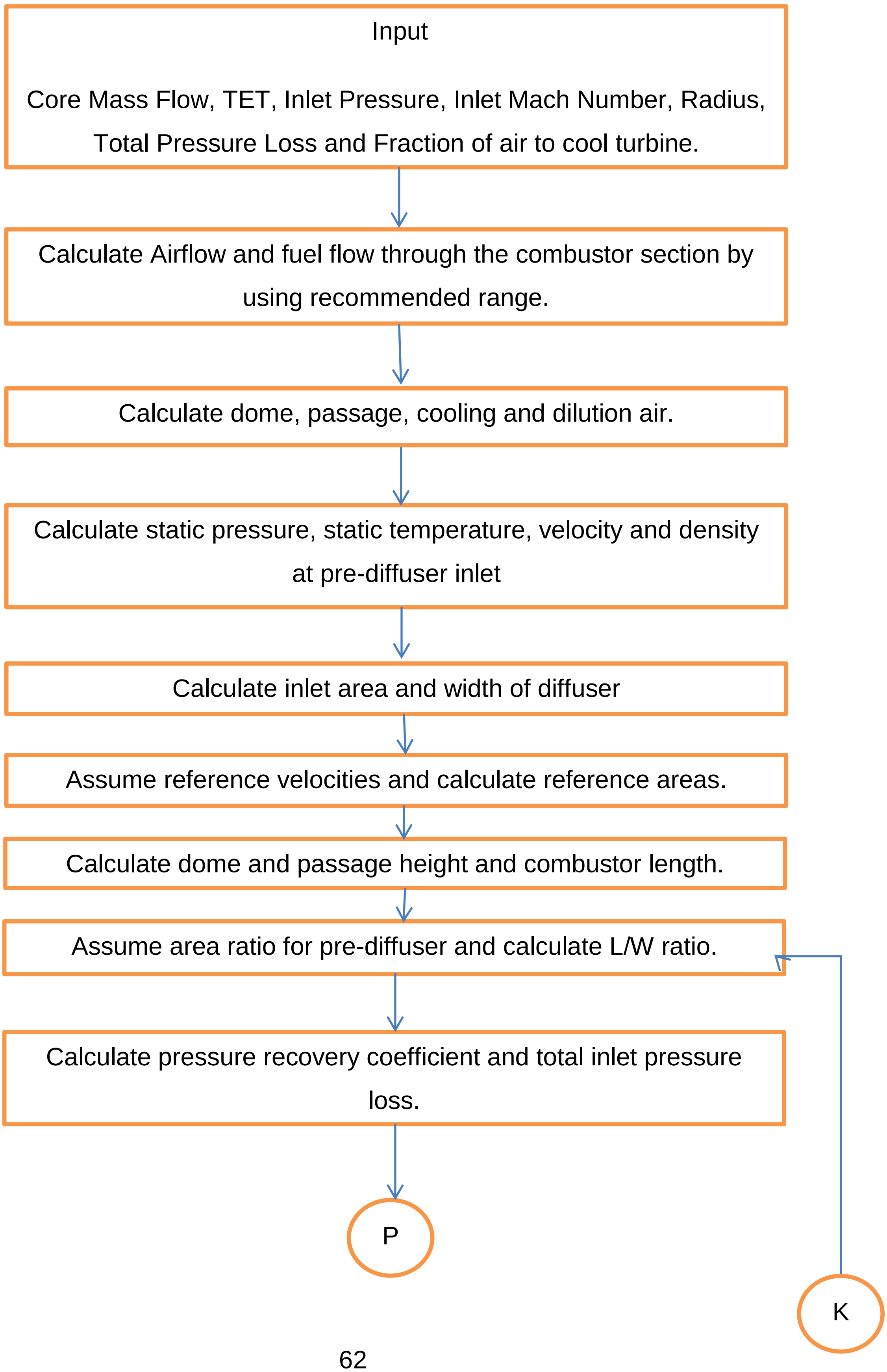
A design methodology of single annular combustor has been presented in this report; verification of this methodology is undertaken in this study from Mohammad [Mohammad, 2009]. Different parameters are required for starting design of single annular combustor, all the parameters required as enlisted below:

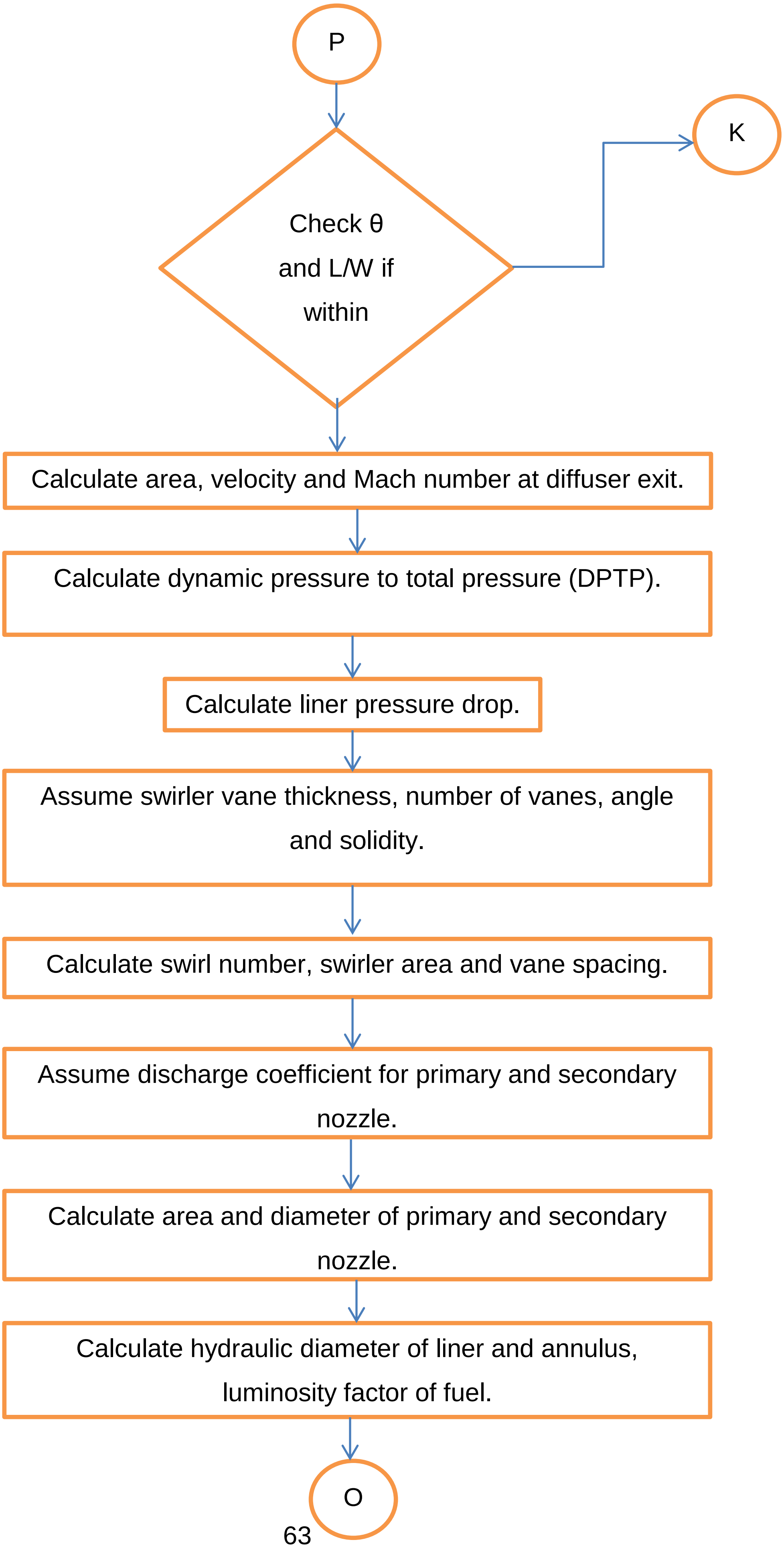
Total pressure at compressor exit (P_{t3}), total temperature at the compressor exit (T_{t3}), total core mass flow rate ($\dot{m}_{\text{total}}$), compressor pitch radius (r_c), fraction of air to cool the turbine ($C_{cooling}$), Mach number at the compressor exit (M_3) and the total pressure loss through the combustor. Constant area combustor is assumed in this study and Jet A fuel is considered for all calculations.

In the first step of this methodology air distribution through combustor is calculated. Airflow through the combustor is divided into two parts, one for dome and other for passage. The airflow through the dome has three different purposes: to atomise the fuel, to stabilize combustion and for dome cooling. The combustor main dimensions are estimated according to the velocity method. That is the dimensions will assume a certain prescribed velocity. Typical velocity ranges are assumed according to conventional combustor designs.

A dual fuel nozzle has been selected and designed for this combustor to satisfy the demand of engine requirements at different missions. Two axial swirlers have been designed for this combustor. Cooling hole has also been designed. Detailed design procedure has been presented in Appendix A. One of the combustor designs is presented in Figure 2-6

Following algorithm shows the basic steps which have been undertaken to design single annular combustor.


Input
Core Mass Flow, TET, Inlet Pressure, Inlet Mach Number, Radius, Total Pressure Loss and Fraction of air to cool turbine.
Calculate Airflow and fuel flow through the combustor section by using recommended range.
Calculate dome, passage, cooling and dilution air.
Calculate static pressure, static temperature, velocity and density at pre-diffuser inlet
Calculate inlet area and width of diffuser
Assume reference velocities and calculate reference areas.
Calculate dome and passage height and combustor length.
Assume area ratio for pre-diffuser and calculate L/W ratio.
Calculate pressure recovery coefficient and total inlet pressure loss.
P
K


P
K
Check θ
and L/W if
within
Calculate area, velocity and Mach number at diffuser exit.
Calculate dynamic pressure to total pressure (DPTP).
Calculate liner pressure drop.
Assume swirler vane thickness, number of vanes, angle
and solidity.
Calculate swirl number, swirler area and vane spacing.
Assume discharge coefficient for primary and secondary
nozzle.
Calculate area and diameter of primary and secondary
nozzle.
Calculate hydraulic diameter of liner and annulus,
luminosity factor of fuel.
O

O

↓

Calculate gas emissivity, temperature in PZ, R1, C1. Some of them by help of NASA CEA.

↓

Calculate discharge coefficient, dilution jet angle, penetration of jets, hole diameter.

↓

Calculate number of holes and number of bands of holes.

↓

Calculate NOx, CO and SMD by help of NASA CEA and empirical correlations.

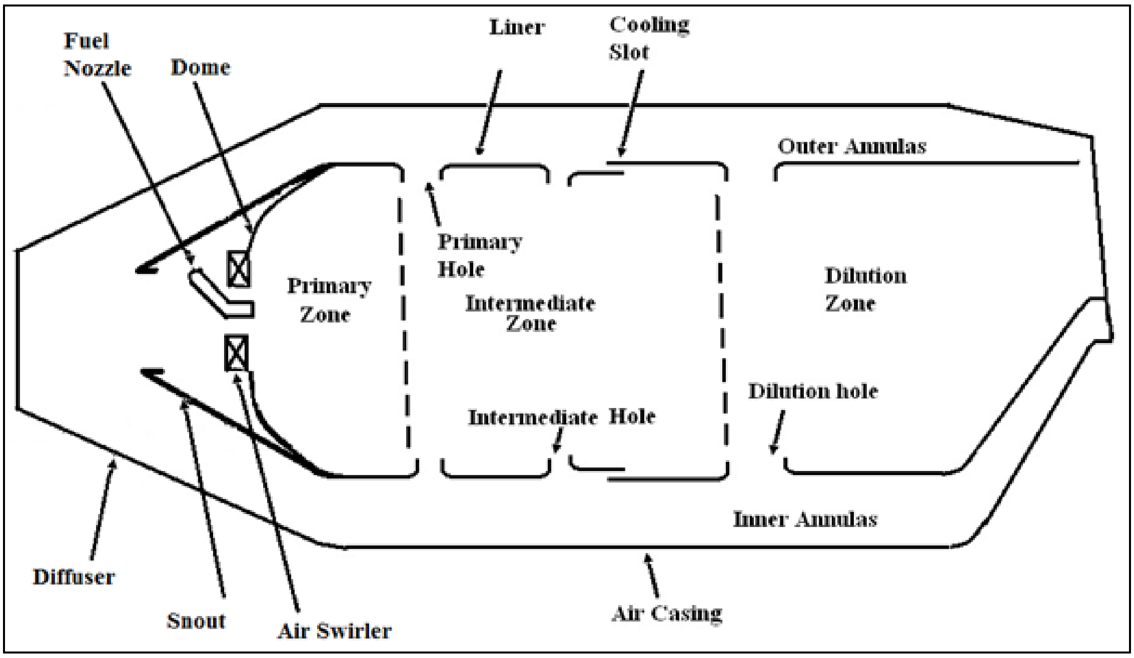


Figure 2-6 Single Annular Combustor [adapted from Lefebvre, 2010]

2.5.2 Reverse Flow Combustor

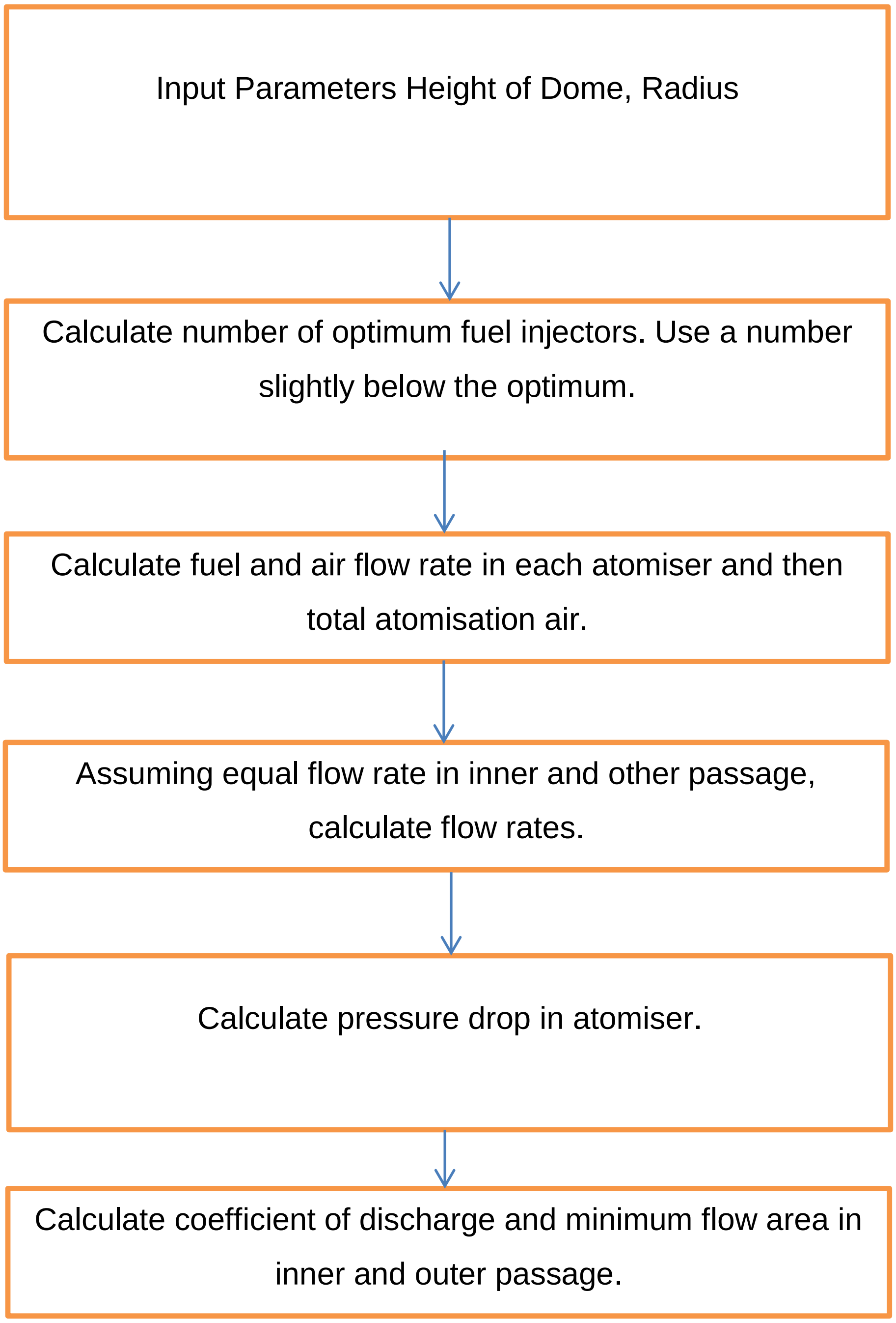
As in case of a reverse flow combustors, the diffuser is usually not designed since radial compressors usually feed the reverse flow combustors. The axial velocity at the exit of the radial compressors is usually low to meet the requirement of the combustors hence a separate diffuser is not necessary unlike the axial flow turbo machines. Reverse flow combustors have more difficulties because of the presence of turn section which does not exist in other combustor configurations [Khandelwal, 2011 (a)]. A study on the design methodology of reverse flow combustor was carried out in this study and the summary of reverse flow combustor design is presented in this section. Detailed design methodology has been presented in Appendix A of this report.

The reverse flow combustors have comparatively higher residence time hence produce higher emissions compared to the axial flow combustors. To minimise the emission by efficient mixing and faster combustion, an advanced air blast atomizers design is studied. In order to shorten the length of the combustor, advanced swirl cup design is studied. This swirl cup consists of a primary swirler, a radial secondary swirler, a venturi and a flare. The application of swirl cup is also expected to achieve high combustion efficiency and good ignition performance etc.

Different cooling schemes such as impingement cooling scheme is used for dome cooling, film cooling scheme for liner cooling and effusion cooling scheme for turn section cooling are studied. The model combustor developed through this design methodology is shown in Figure 2-7. The design of primary holes and dilution holes is also examined, including the type of the holes, the number of the holes and the hole space ratio, etc.

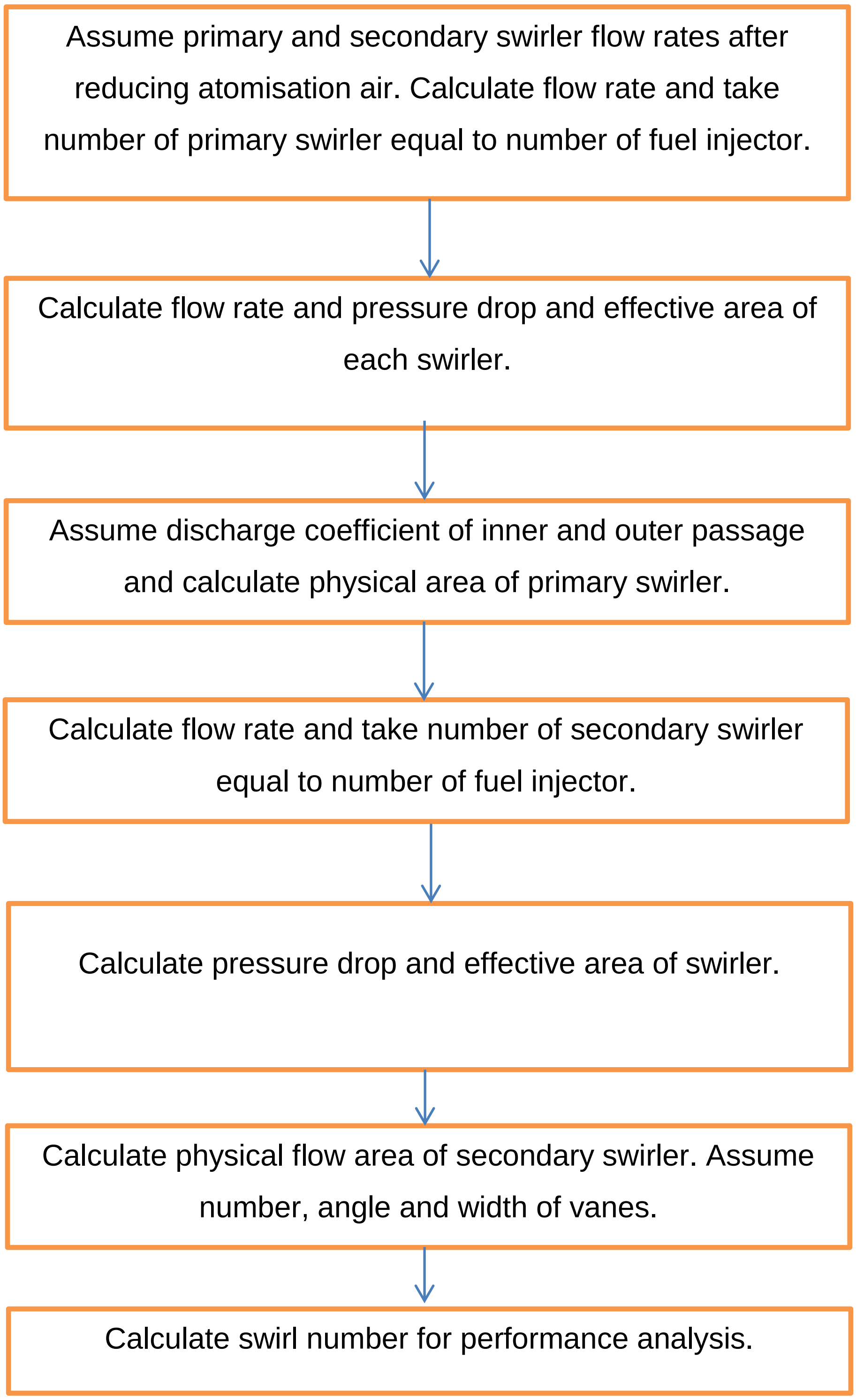
Following algorithm shows the design methodology adopted for designing different parts of reverse flow combustor. This methodology has resulted into substantial contribution to knowledge. It is being used in several research projects in the university, after its publication [Khandelwal, 2011 (a)].

Fuel Injector design.



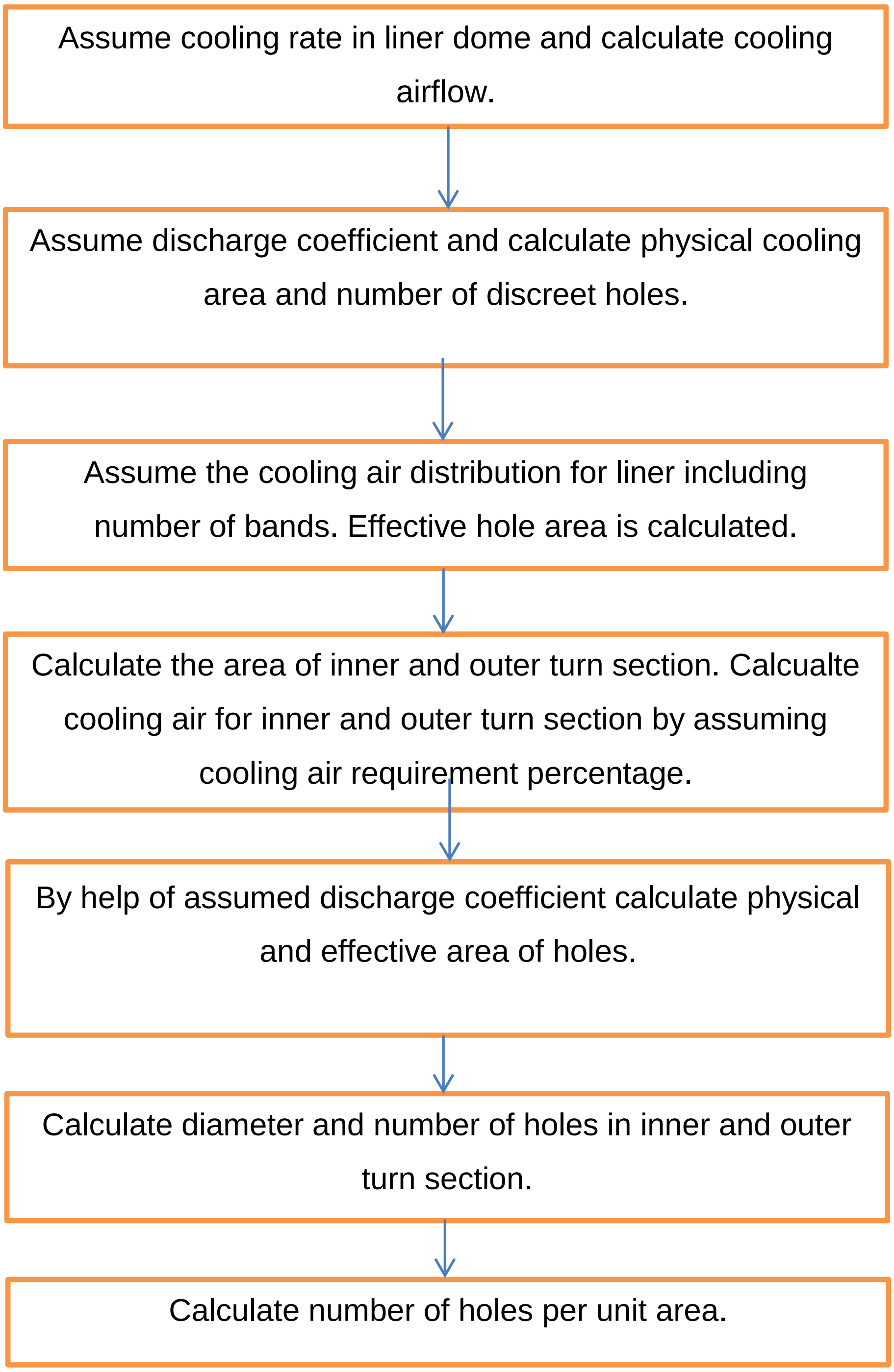
Swirler design

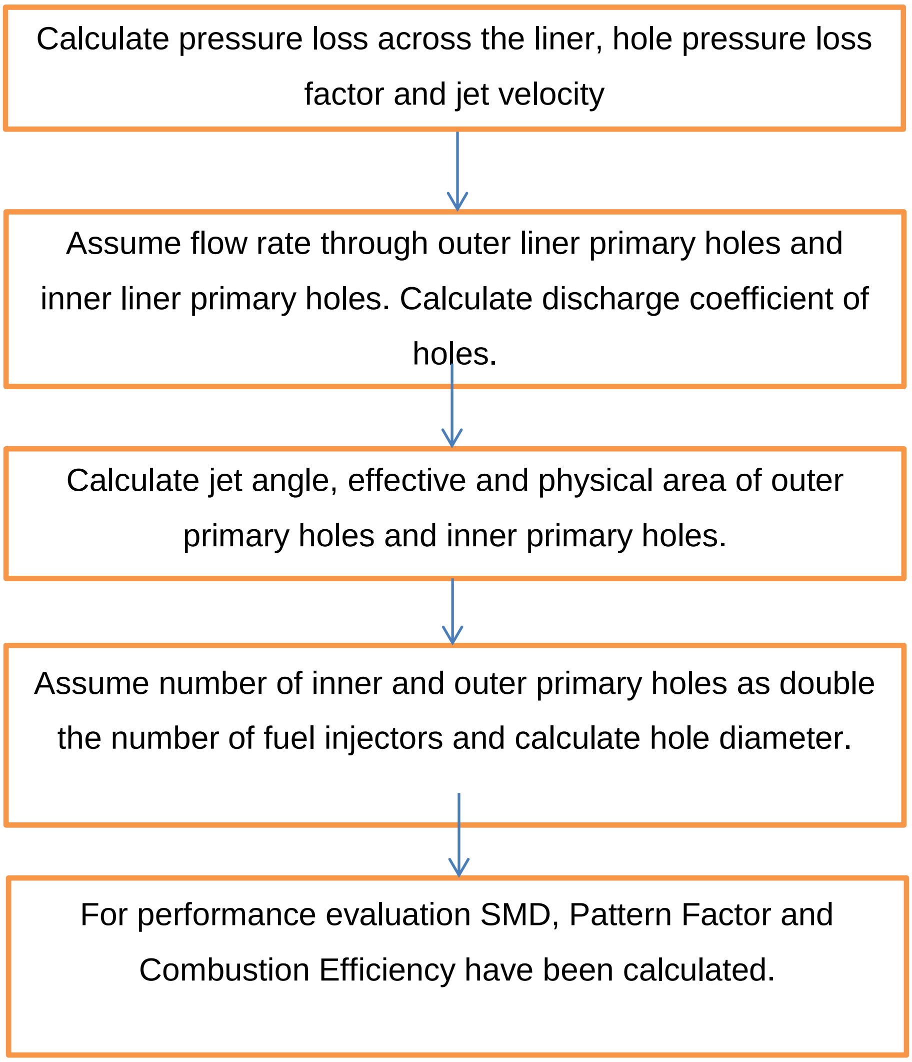
Double swirl cup design is selected for shorter combustor length.



Cooling design

Impingement film cooling is selected to cool the liner dome. Film cooling is selected for cooling of liner.




Calculate pressure loss across the liner, hole pressure loss factor and jet velocity
Assume flow rate through outer liner primary holes and inner liner primary holes. Calculate discharge coefficient of holes.
Calculate jet angle, effective and physical area of outer primary holes and inner primary holes.
Assume number of inner and outer primary holes as double the number of fuel injectors and calculate hole diameter.
For performance evaluation SMD, Pattern Factor and Combustion Efficiency have been calculated.

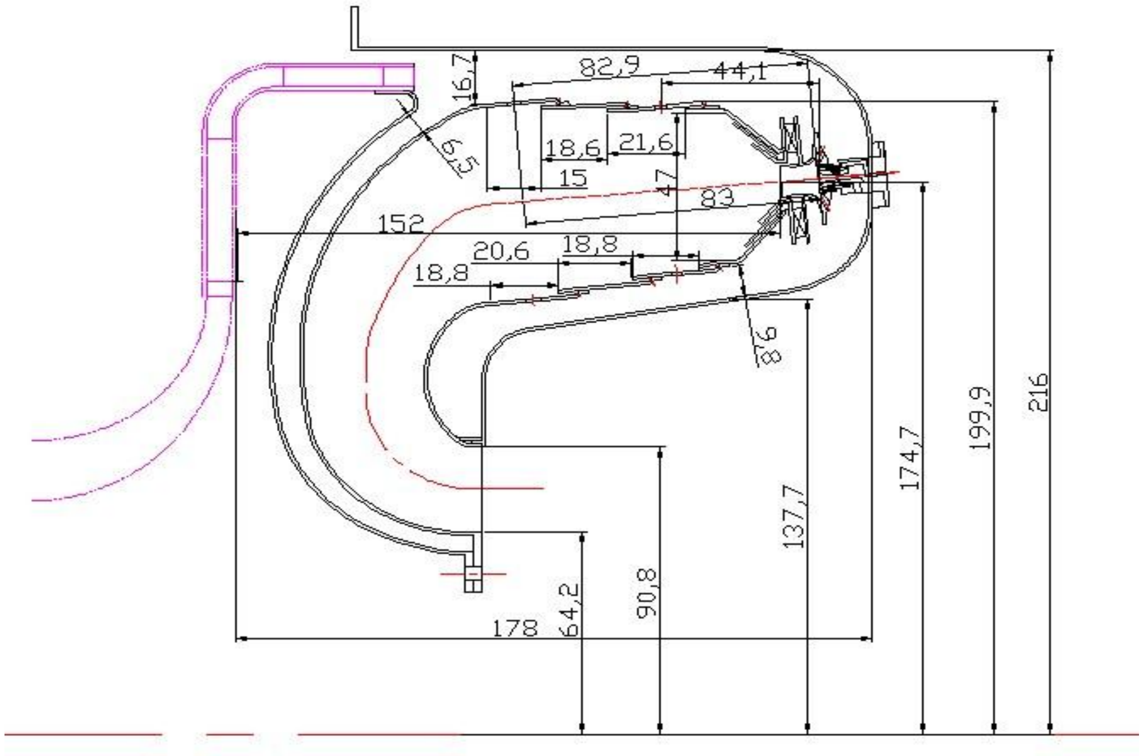


Figure 2-7 Developed model Combustor schematic diagram [Khandelwal, 2011 (a)]

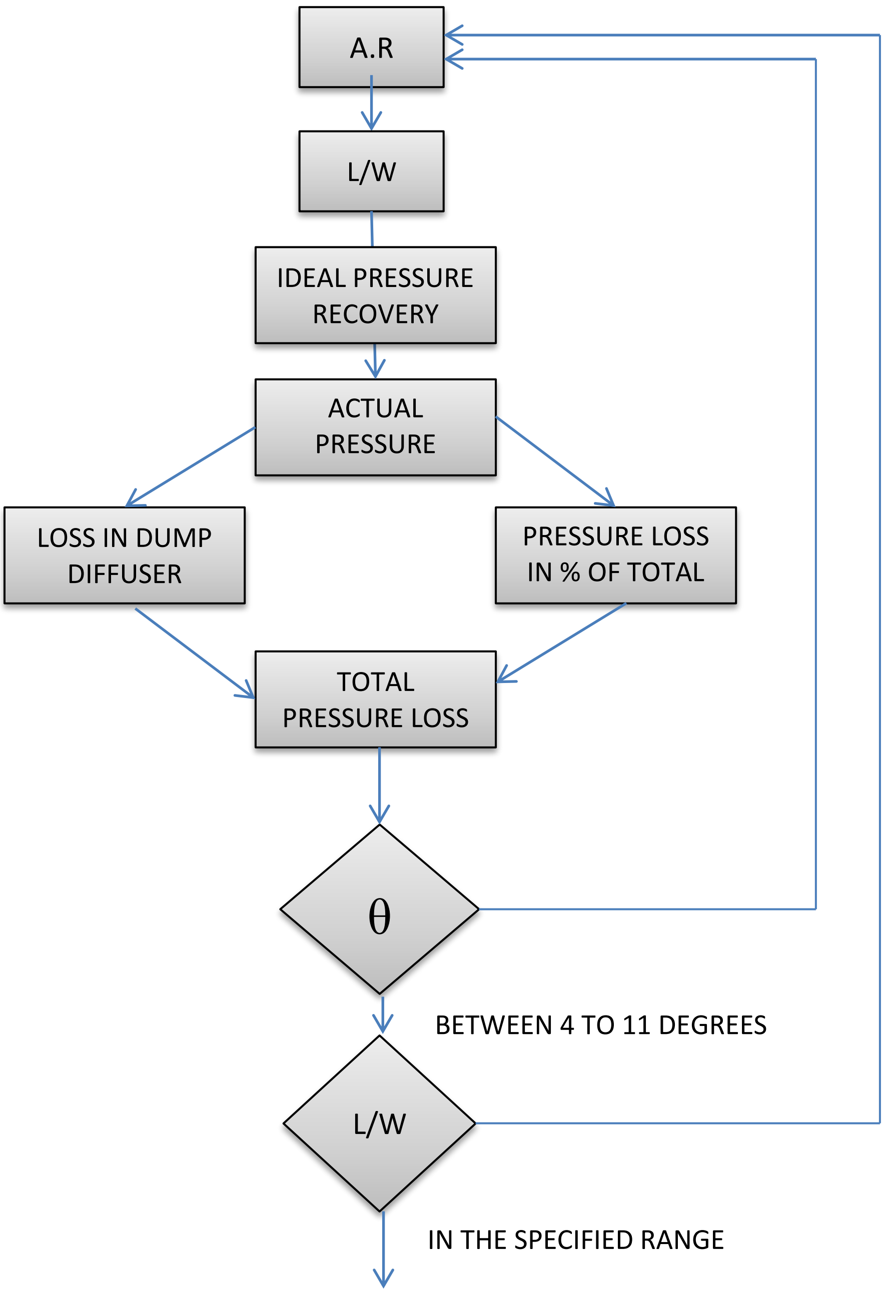
2.5.3 Radially Staged Dual Annular Combustor

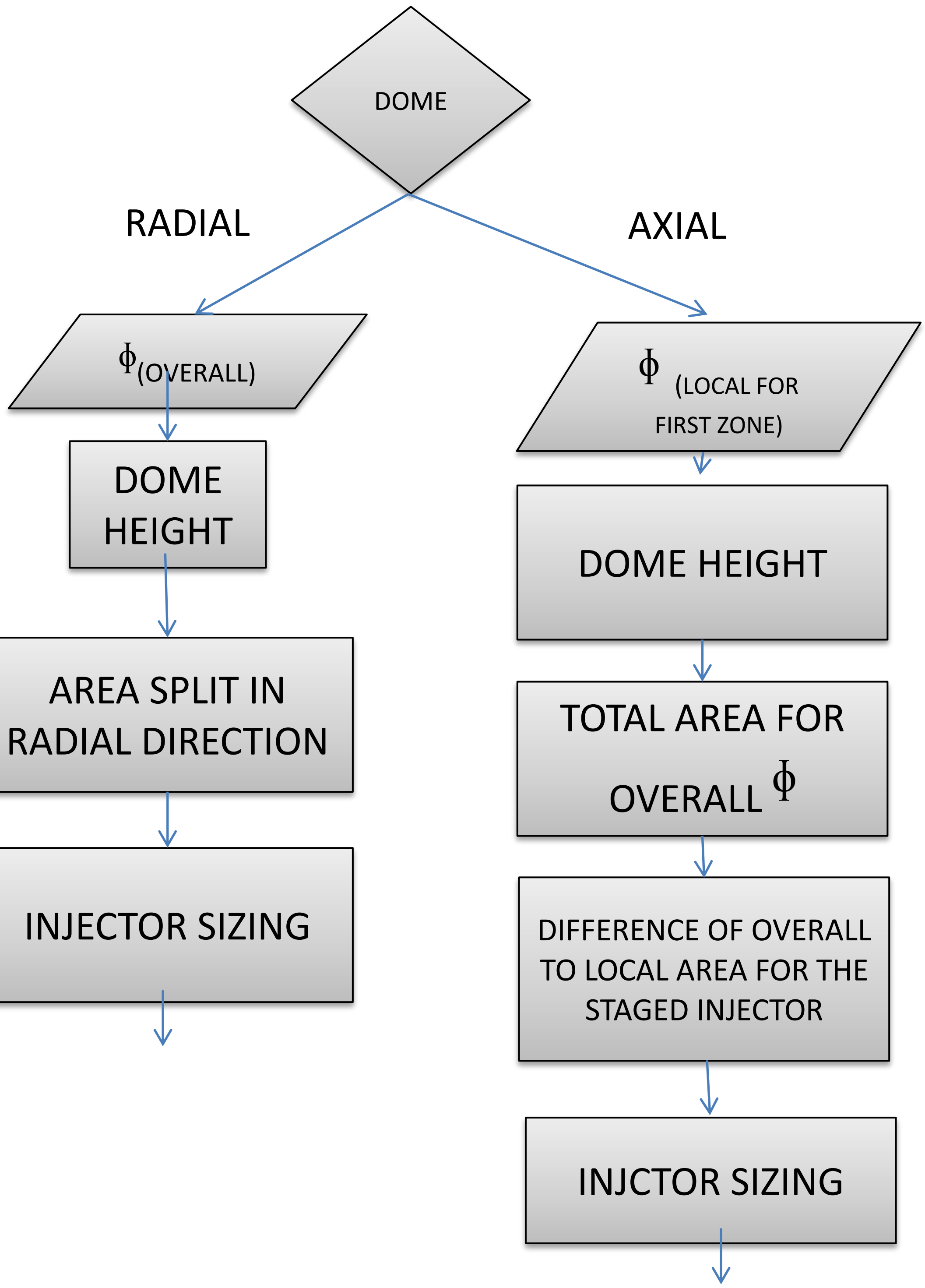
The design methodology developed in this study follows the conventional method to certain extent and then the idea is extended to accommodate the second stage while keeping in mind the principal changes that follow in terms of change in the combustor cross sectional area, the overall equivalence ratio for low emissions and also the placement of swirlers and the number of fuel injectors [Khandelwal, 2012 (a)]. The design splits the combustor in two parts and each part is individually designed and assembled to form the final design. The assumptions made in designing the combustor based on the methodology developed are on the basis of existing dual annular combustor from GE-90 engine.

The design methodology developed in this project follows a two-fold approach. The first step is overall combustor sizing. The next step is detailed design of components and zones. The basic combustor sizing follows the classic combustor design methodology and this forms the overall dimensions of the combustor. The compressor exit flow conditions and the geometrical conditions are taken as the input for calculation, γ, dome and passage velocities are assumed.

Because the dump diffuser is shorter and lighter, not sensitive to the inlet conditions, manufacture tolerance and so on, most of the modern combustors use dump diffuser. In the design methodology developed for the model combustor, a single pass axial swirler is incorporated. The typical parameters for axial swirler design include: the vane angle θ, should be 30~60º; the vane thickness should be 0.7~1.5mm and the number of vanes should be 8~16. The swirler design developed in this work is same for both the main and the pilot stage. The design of swirler needs many empirical data or they must be assumed. There are two variations of swirlers considered for the radially staged dual annular combustor model. The pilot swirler is taken to be having the outer diameter equalling to 75% of the outer diameter of the main swirler. This consideration is based on the assumption that the pilot burner works at lower power with lower fuel flow rates, hence to maintain the equivalence ratio in the stability range, the amount of air admitted should be lesser.

The functions and Requirements of the dilution holes are to be discussed and understood before committing for their design. The dilution zone is use to achieve a desired temperature distribution required by turbine components. The quality of temperature distribution is generally measured by “pattern factor (PF)”. One of the problems in the design of the dilution holes is that the designer is given two options or methods to choose from. The dilution hole sizing can be designed by two different methods, NASA design method and Cranfield method. This project follows the Cranfield design method of designing the dilution holes. Conventional film cooling scheme is chosen as the cooling scheme of liners. The main difference between these two methods is that Cranfield method stresses on the hole size while the NASA method emphasises on the hole spacing. Thus, for any given value of J- momentum flux ratio and downstream distance, the Cranfield method leads to an optimum hole size and the spacing between them is then calculated based on the number of holes. On the other hand, NASA method first identifies the optimum hole spacing and the hole size is then determined based on the number of holes. The two different approaches produce different dilution hole designs and the final selection is always the designer's choice. Detailed design procedure of dual annular radial combustor is presented in Appendix A [Khandelwal, 2012 (a)]. Algorithm below shows the preliminary design steps of diffuser and dome of double annular axially and radially staged combustor.




DOME
RADIAL
AXIAL
ϕ(OVERALL)
ϕ (LOCAL FOR FIRST ZONE)
DOME HEIGHT
DOME HEIGHT
AREA SPLIT IN RADIAL DIRECTION
TOTAL AREA FOR OVERALL ϕ
INJECTOR SIZING
DIFFERENCE OF OVERALL TO LOCAL AREA FOR THE STAGED INJECTOR
INJCTOR SIZING

Overall dimensions obtained by preliminary design of dual annular radial combustor are presented in Table 2-2.

Table 2-2 Overall dimensions of the model combustor

Overall Combustor Dimensions	
Combustor Inlet Diameter (m)	0.357
Combustor Outlet Diameter (m)	0.38
Length of the diffuser (m)	0.157
Liner Length/Dome Height (m)	1.5
Length of Liner (m)	0.116
Dome Height (m)	0.077
Distance between diffuser outlet and dome (m)	0.094
Swirler Length (m)	0.012
Swirler Outer Diameter (m)	0.039
Swirler Inner Diameter (m)	0.01
Swirler Vane Angle	45
Swirler Vane Thickness (m)	0.002
Length of Dilution Zone (m)	0.088
Total Length of Combustor (m)	0.368

2.5.4 Axially Staged Dual Annular Combustor

In a series or axially staged combustor, a portion of fuel is injected in to a conventional primary zone. Additional fuel usually premixed with air is injected downstream in to the main combustion zone. The main combustion zone operates at lower equivalence ratios to minimize the formation of NO_x and smoke. The primary combustion zone is used on engine start up and provides the entire necessary temperature rise needed to raise the rotational speed up to engine idle conditions. At higher power settings, the main combustion zone is brought to usage. As the engine power raises the function of primary zone shifts from operational importance to the emission control functions. That is at higher power, the primary zone supplies the heat needed for rapid combustion in the main zone. This enables the designer to optimise the main zone for very little residence time.

The overall design method developed for axial staging does not differ much in comparison to the parallel staged dual annular combustor. However, changes are made in the detailed design of components. The important changes made are the determination of axial spacing between the stages, the consideration of inlet dome cross section area for a single injector. An important assumption on axial spacing between the stages is made based on the available data and discussion with the industrial experts. The spacing is assumed to be the length of the primary zone length of the pilot combustor. The pilot zone is assumed to be operating fuel rich to facilitate easy light up, good combustion stability and a reliable altitude relight [Khandelwal, 2012 (a)]. Much of the design methodology for axially staged dual annular combustor remains similar to that of the radial staged dual annular combustor. The important change is observed only in the design of casing and liner section of the combustor. Due to axial staging, the combustor length increases and the important assumption is made for the determination of axial stage spacing. The spacing is taken as the primary zone length of the pilot burner. This results in staging the design method itself. The liner two as it is termed here is the main burner liner.

Development of design methodology for axially staged dual annular combustor is in final stage and will be added in the Appendix A [Khandelwal, 2012 (a)].

2.5.5 Radially Staged Triple Annular Combustor

A triple annular dry low emission industrial combustor is designed as a part of this project, which is similar with LM6000 combustor. Advanced DLE methods such as lean fuel combustion concept, air fuel premix method, staged combustion concept, triple annular arrangement, multi-passage diffuser design, double wall cooling system, double annular counter rotating swirlers (DACRS) and heat shield are employed to reach the aim of decreasing pollutant emissions.

Firstly, the diffuser is designed as multi-passage diffuser which consists of four passage divided by three splitter vanes. The pressure loss of this diffuser is about 0.99%. Secondly, the casing and liner are sized by velocity method based on the operating conditions of GDP [2010] combustor and equivalence ratio is selected as 0.6. The volume of combustor is nearly two times as that of last GDP [2010] aero combustor. Next, the flow area of fuel injector is calculated after air flow through it has been calculated. Because the fuel injector is DACRS and it works as a combination of fuel injector and swirler, conventional swirler design approach is employed as the design method for calculating dimensions. Meanwhile, the diameter of fuel injection holes have been selected as 0.6 mm based on recommended data and each outer vane has three injection holes. For staged combustion, the DACRS have been arranged in three. Then, cooling system is designed both in dome cooling and liner cooling. Heat shield is adopted between fuel injectors to cooling the dome area and four rings of cooling holes are designed at the dome plate. Double wall cooling method is employed to reduce the temperature at liners. Both the diameters of dome cooling and liner cooling holes are based on the correlation among flow area, cooling air flow rate, air density and pressure drop. Finally, the dilution holes are calculated by Cranfield method.

Detailed design procedure of radially staged triple annular combustor is presented in Appendix A of this report. Figure 2-8 shows basic dimensions of the combustor which have been designed.

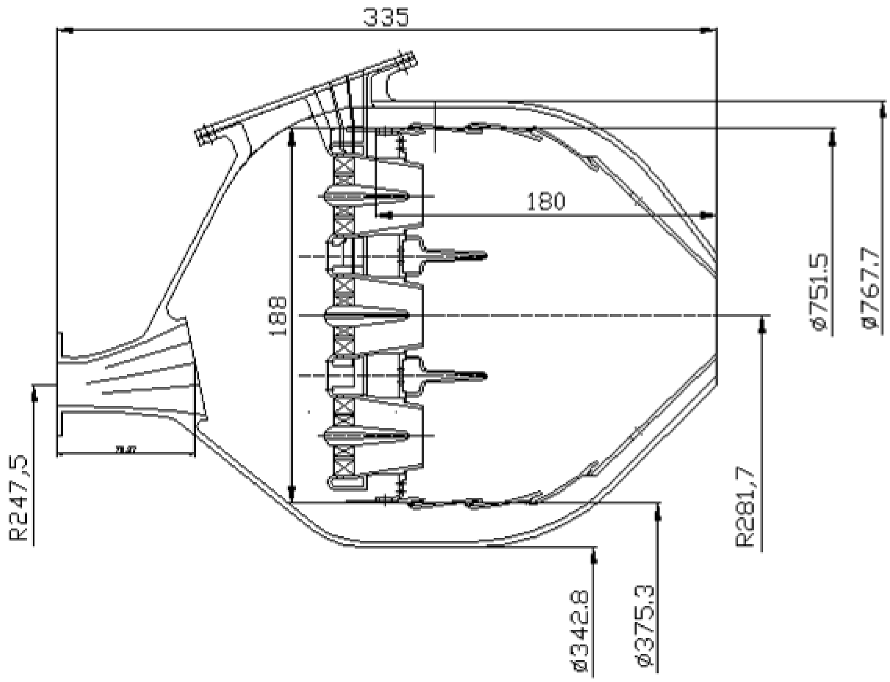


Figure 2-8 Design Results of Combustor [Khandelwal, 2012 (f)]

2.6 Conclusions

Design methodologies for different combustors have been developed based on empirical approach. The combustors for which design methodologies have been developed are single annular combustor, double annular radial and axial combustor, triple annular radial and axial combustor and reverse flow combustor. Combustor design methodology includes detailed designing of different parts of combustors which includes diffuser, fuel injector, swirler, casing, liner, cooling holes and other relevant parts. Design methodologies developed are resulting in combustors of reasonable resemblance with present combustors. Estimated performance analysis also shows that the combustors designed are giving appropriate performance.

Developed design methodologies would substantially add contribution to knowledge as there are no such design methodologies available in public domain.

3 DILUTION ZONE LESS NOVEL COMBUSTOR

3.1 Abstract

An innovative combustor concept has been proposed in this study. The proposed concept involves eliminating a dilution zone of a gas turbine combustor and introducing the dilution air by help of NGV for maintaining the right temperature traverse for turbine blade. This concept would help in decreasing the length of combustor, which would eventually lead to decrease in weight, emissions, fuel consumption and vibration. Numerically it has been shown that such a system is physically feasible. The proposed combustor is economically competitive which can meet the requirements. A patent has been filed for this concept and patent search shows that a patent on this design could be granted. Future work includes experimentally testing of proposed NGV at operating conditions of an engine. This concept would lead to contribution to knowledge in field of combustors.

3.2 Introduction

The two main governing parameters besides the turbine & compressor efficiency are the ratio of highest to lowest cycle temperature and the compression ratio. This has seen the increase in overall efficiency of the gas turbine engine. As a result, turbine inlet temperatures and compression ratios have increased dramatically in the last decades, but the major limiting factor in the increase of the turbine entry temperature until early sixties was the unavailability of materials that could withstand the thermal and mechanical loads. Even though there have been significant advances in the materials and material properties, still unable to achieve those TET which could extract the best performance of the engine. Consequently other alternatives have been explored into in-order to achieve better performance. Hence new improved cooling technique, such as internal cooling using impinging jets or simple convective cooling have been the source of extended range of feasible turbine inlet temperatures. Since aircraft engines of the future demand ever-increasing performance levels, higher turbine inlet temperatures, higher thrust-to-weight ratios, and maximum thermal efficiencies, turbine-related heat transfer issues and its accuracy of prediction are becoming more critical to gas-turbine research and design. Hence concepts including internal cooling passages, external film cooling, high-tech ceramic materials, and thermal barrier coatings, to name a few, have all been implemented by industry in an effort to combat the unfavourable effects of excessive surface heat transfer. But that's not all, aspects related to the environment such as global warming or the shortage of natural resources have also identified a need to further optimise the efficiency of gas turbine engines. These and economic issues create a continuous force on the aero engine and the gas turbine industry towards lighter, more efficient, cleaner and cheaper products. Increase in parameters such as inlet temperature and pressure ratio have been the key points in reducing specific fuel consumption, increase in the power output, rise in overall engine efficiency and achieving a reduction in CO_2 emissions.

Numerous research work on NGV's has been carried out to evaluate the effects of Reynolds number, free stream turbulence, acceleration, transition, and surface roughness on the blade heat transfer. Film-cooling of the NGV & turbine blades has been an extensively researched topic for over the last 30 to 35 years [William, 2005; Kercher, 2003].

In depth study on variation of every possible geometrical parameter; including surface angle, entrance length, hole spacing, compound angle, lateral expansion angle, forward expansion angle, area ratio, and multiple row configurations where carried out and checked for their relevant effects on turbulence intensity, pressure gradient, and the state of the approaching boundary layer. An excellent review of the relevant shaped hole literature, which primarily focused on flat-plate studies, was given by Bunker [Bunker, 2005]. Some studies have presented results for partially and/or fully cooled nozzle guide vanes, but the deficiency of many of those studies is the lack of high resolution effectiveness measurements. Studies involving a single row of fan-shaped cooling holes on a vane surface have been performed by Zhang et al. [1999], Zhang and Pudupatty [2000], and Colban et al. [2005].

Studies on effectiveness measurements were made with fan-shaped holes on the suction side by Zhang et al. [1999] and on the pressure side by Zhang and Pudupatty [2000]. Results indicated an increase in effectiveness on the suction side for the blowing ratio range from 0.5 to 1.5 and a decrease in effectiveness on the pressure side for the blowing ratio range from 1.5 to 2.5. Colban et al. [2005] presented adiabatic effectiveness measurements for eight single rows of fan-shaped holes on both the pressure and suction sides. Their results indicated that in regions of high convex curvature, particularly on the suction side near the leading edge, jet lift-off was prevalent, and increased with blowing ratio. Colban et al. [2005] also noted a decrease in effectiveness with increased blowing on the pressure side, which was attributed to partial jet lift-off and hot gas entrainment.

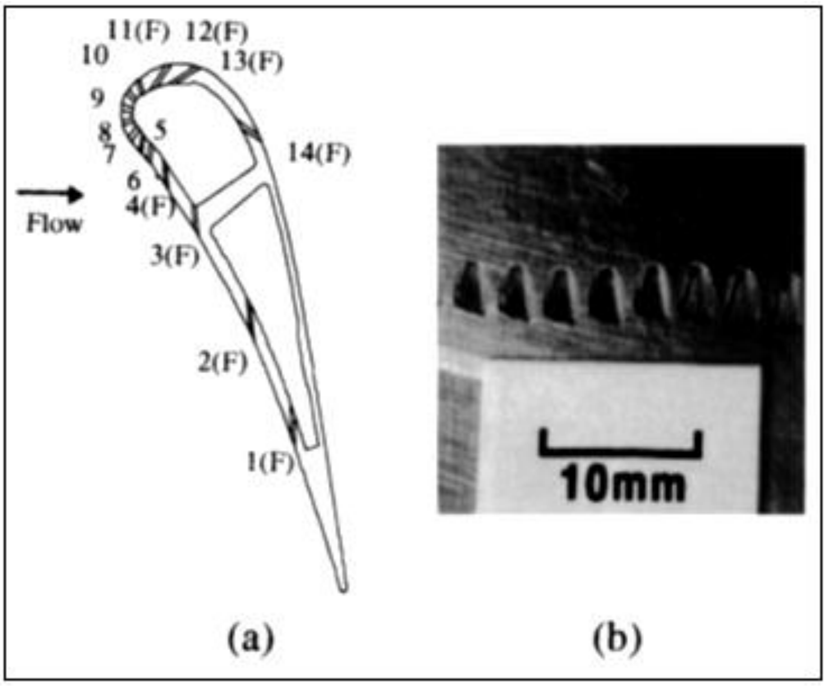


Figure 3-1 (a) Cross-section of NGV with full film cooling geometry, holes marked F are fin shaped holes. (b) Fan-shaped holes [Guo, 1998].

Effectiveness measurements were made by Guo et al. [1998] in a transonic facility for a turbine airfoil with multiple rows of fan-shaped holes as shown in Figure 3-1. Results showed higher values of effectiveness for fan-shaped holes than for cylindrical holes. However, the decay in effectiveness on the pressure side was faster for fan-shaped than for cylindrical holes, which was most such as the result of a better lateral coverage for the fan-shaped holes.

Langston [1977] was one of the first for presenting descriptions of endwall secondary flow considered in a gas turbine. The incoming boundary layer on the endwall has a constant static pressure profile in the span wise (normal to wall, along span of airfoil) direction as shows in Figure 3-2. However, the boundary layer has a non-uniform velocity profile because of the difference in velocity between fluid entrained near the wall and fluid in the mainstream, which corresponds to a non-uniform total pressure profile.

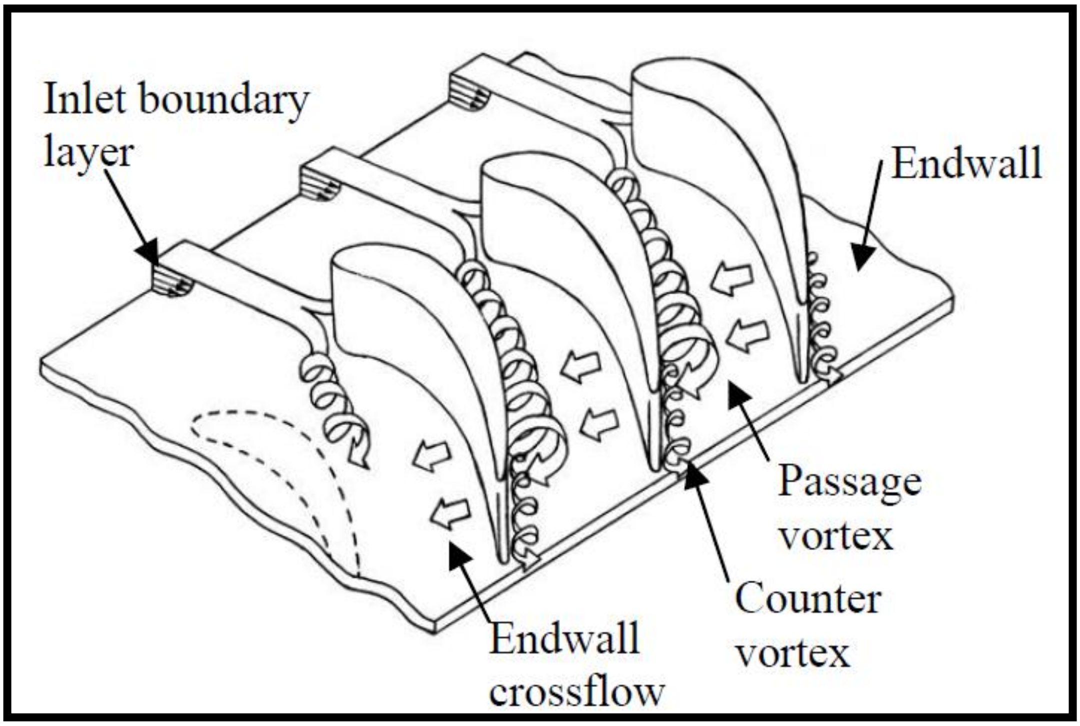


Figure 3-2 Secondary flow model presented by Langston [1977].

Langston also observed that as the boundary layer stagnates on the airfoil, the total pressure profile becomes a span wise pressure gradient that drives the flow to the endwall. This turning creates a vortex that splits at the stagnation point and wraps into two legs around the pressure and suction sides of the turbine airfoil—this is the horseshoe vortex. The portion of the horseshoe vortex on the pressure side, known as the passage vortex, is augmented further by the inherent pressure gradient between airfoils (i.e., the pressure side of one airfoil faces the suction side of its neighbour). The portion of the horseshoe vortex that passes to the suction side, known as the counter vortex, has an opposite sense of rotation to the passage vortex, and tends to orbit the passage vortex as it interacts with the passage vortex downstream of the stagnation point.

A series of experiments have been reported for various injection schemes upstream of a nozzle guide vane with a contoured endwall by Burd and Simon [2000], Oke, et al. [2000] and Oke et al. [2001]. In their studies they have shown that the coolant was injected from an interrupted, flush slot that was inclined at 45° just upstream of their vane. They found that most of the slot coolant was directed toward the suction side at low slot flow conditions. As they increased the percentage of slot flow to 3.2% of the exit flow, their measurements indicated better coverage occurred between the airfoils. In contrast, the study by Oke et al. [2001] used a double row of film cooling holes that were aligned with the flow direction and inclined at 45° with respect to the surface while maintaining nearly the same optimum 3% bleed flow of their previously described studies. They found that the jets lifted off the surface producing more mixing thereby resulting in a poorer thermal performance than the single slot. Roy et al. [2000] compared their experimental measurements and computational predictions for a flush cooling slot that extended over only a portion of the pitch directly in front of the vane stagnation. The adiabatic effectiveness measurements indicated that the coolant migrated toward the pressure side of the vane. Their measurements indicated reduced values of local heat transfer coefficients at the leading edge when slot cooling was present relative to no slot cooling.

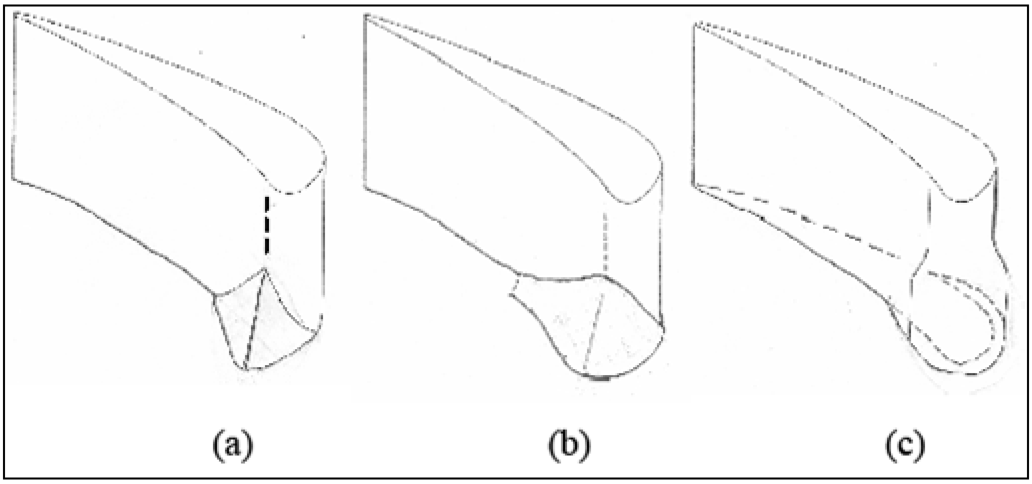


Figure 3-3 Basic leading-edge fillet geometries. (a) Sharp/pointed. (b) Rounded. (c) Bulb-type.

Colban et al. [2002(a); 2002(b)] reported flow field and endwall effectiveness contours for a backward-facing slot with several different coolant exit conditions. Their results indicated the presence of a tertiary vortex that developed in the vane passage due to a peaked total pressure profile in the near-wall region. For all of the conditions simulated, the effectiveness contours indicated the coolant from the slot was swept towards the suction surface.

Past research has shown that modifications to the leading edge of a gas turbine vane can reduce or eliminate some of the features of the secondary flow. Sauer et al. [Sauer, 2000] used an asymmetric leading edge bulb (shown in Figure 3-4) to intensify the counter vortex, which resulted in a 50% reduction in aerodynamic losses at the exit of the vane passage. Zess and Thole [2002] used computational fluid dynamics (CFD) to design an asymmetric leading edge fillet, which they later experimentally tested. Their research indicated elimination of the leading edge horseshoe vortex and an order of magnitude reduction in turbulent kinetic energy levels associated with vortex development. Becz et al. [2004] studied two bulb designs as well as an asymmetric elliptical fillet and found that only the fillet reduced overall total pressure loss. It saw a slight reduction in airflow turning, which agrees with the wall shear stress results obtained by Stephen Lynch [2008]. Other studies have considered the effects of leading-edge modifications on the thermal environment of the vane or blade.

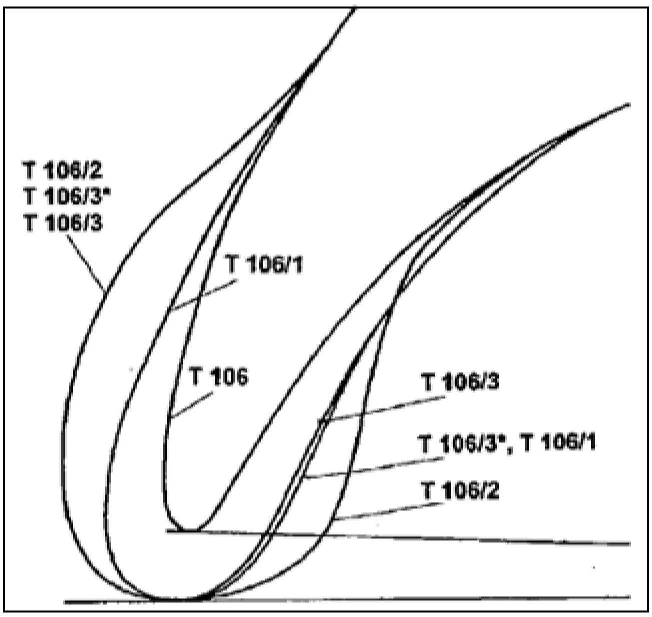


Figure 3-4 Configurations of Leading Edge endwall bulb configurations [Sauer, 2000].

Shih and Lin [Shih, 2002] performed computational studies on two fillet designs with and without swirl in the incoming flow profile. Their study indicated the surface heat transfer and aerodynamic loss on the airfoil and endwall decreases with leading-edge fillet and inlet swirl. Mahmood et al. [2005] performed smoke flow visualization, took total pressure measurements, and obtained Nusselt number distributions on the endwall for four fillet geometries. Their results indicated a reduction in the leading edge vortex size and lower endwall heat transfer coefficients for all fillet geometries, with a concave elliptical geometry showing the largest reduction in heat transfer.

Increasing a step further towards advanced combustors an innovative concept of combustor has been proposed in this study. According to the proposed concept, dilution zone would be eliminated form gas turbine combustors, and dilution air would be introduced by the NGV for maintaining the desired temperature traverse profile. This concept would help in decreasing the length of combustor, consequently weight, emissions, fuel consumption and vibration.

3.3 Methodology

The invention relates to elimination of the combustor dilution zone and using the innovative NGV which is integrated within the combustor as shown in Figure 3-5, a major source through which the dilution air is being injected into the combustion chamber. This conceptual NGV design is said to carry out the dilution of the hot gas stream coming from the combustor primary zone. The concept of placing the dilution holes on the leading edge of the NGV has been considered and the cooling air for dilution of the hot gas stream is being taken from the bleed of the compressor exit. The main idea behind this concept is to reduce the length of the combustor by eliminating the dilution zone of the combustor and using the NGV as the source of injecting the dilution air into the main gas stream. Hence doing so, we reduce the temperature and achieve efficient mixing of the dilution air and combustion air.

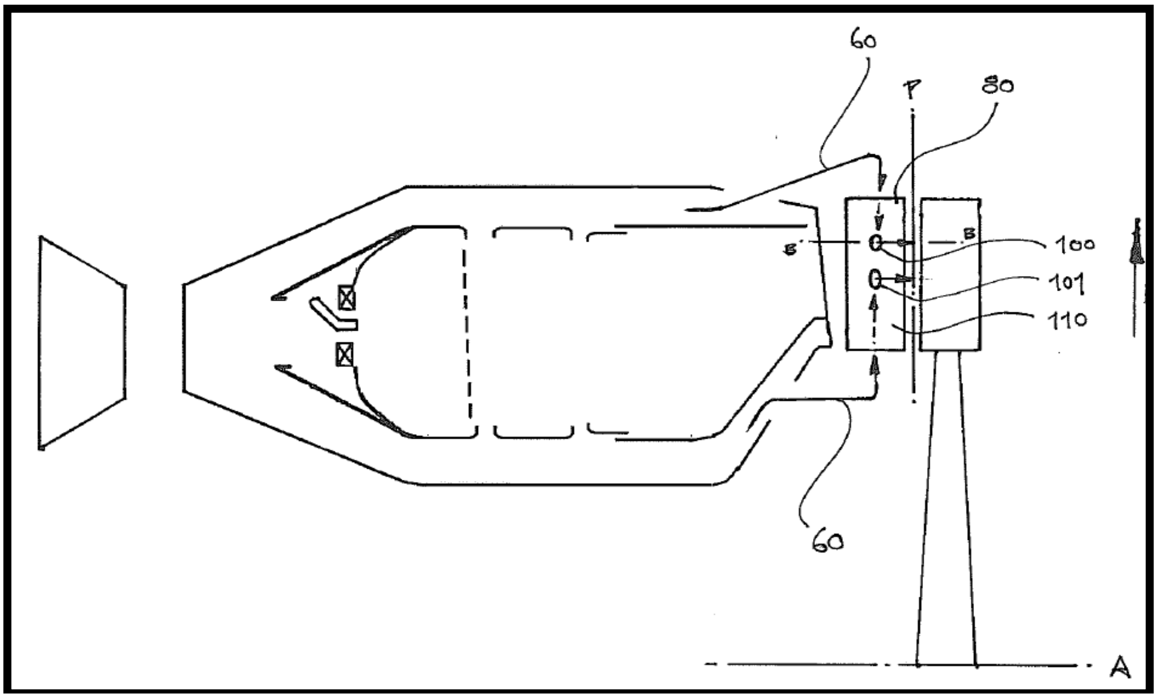


Figure 3-5 Sectional view of dilution-zone less combustor including conceptual NGV

This concept allows us to achieve uniform temperature profile at the exit of the NGV. Also higher mass flow through NGV blades eliminates the need for complex internal machining usually which is required for cooling passages. This is because, as the velocity and mass flow increase, convection between the NGV surface and air also increases. Reduced length and cooling requirements have positive effects on the combustor material selection and manufacturing contributing to simplicity and economy in general. Taking the example of an axially staged annular combustor and introducing this conceptual idea of NGV, then the elimination of the tertiary region of such a combustor could give us a advantage of shortening the combustor length to around 40% of its total length and whole engine length by 40%. This is an extreme case. If a conventional or a radial staged combustor is to be modified, then about 50-60% of the total liner length is eliminated. This equates to great savings in the system weight and complex requirements.

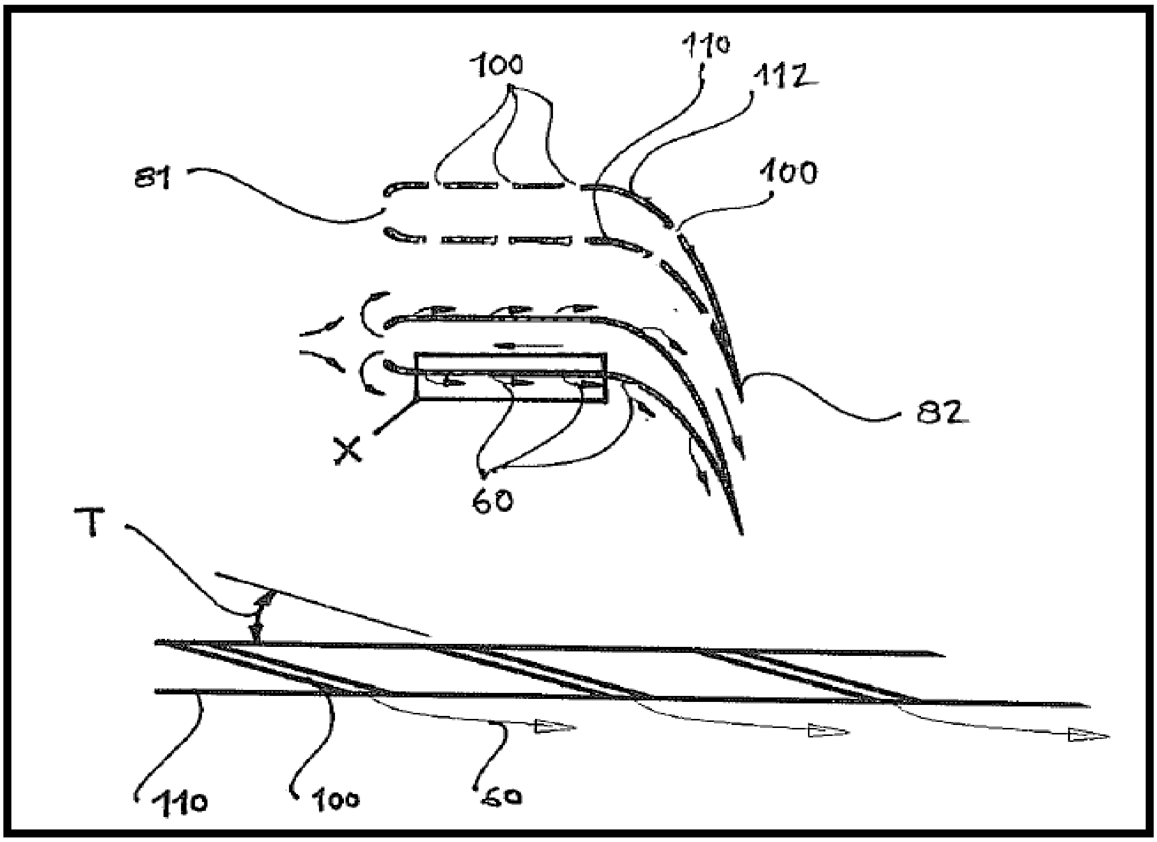


Figure 3-6, Sectional view of conceptual NGV

Figure 3-7 One way of injecting air though NGV

Figure 3-7 shows one of the many explorable ways of injecting air out from the surface of the NGV. The air is preferably injected tangentially to avoid any flow disturbance keeping the effectiveness of the NGV intact.

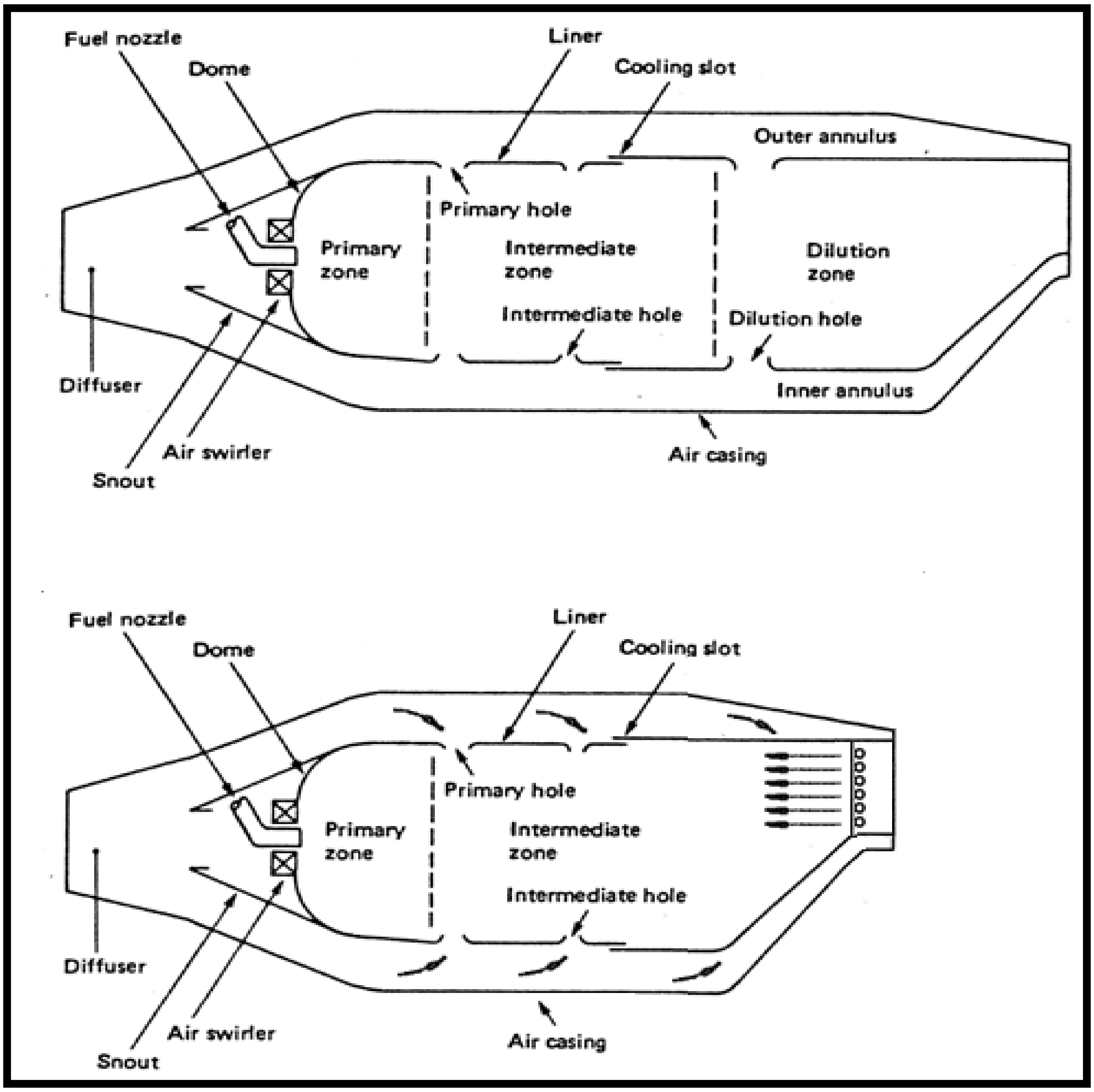


Figure 3-8, Comparison between conventional and conceptual combustor

Figure 3-8, shows the difference between the conventional combustor design and modified combustor with the conceptual integrated NGV design. Here the dilution zone of the conventional combustor is eliminated and the conceptual integrated NGV is made to act as the dilution zone for the modified combustor. Hence doing so it can be seen that there is a reduction in combustor length due to dilution zone elimination and makes the engine shorter and lighter when compared to the earlier design. A new concept for gas turbine combustors (for both kerosene and hydrogen fuelled combustors) is proposed. According to the new concept a substantial amount of dilution air is introduced into the turbine nozzle guide vanes directly, in a controlled manner, to ensure optimum turbine blade temperature traverses are maintained. Detailed patent search and their results have been included in further sub-section.

3.4 Calculation for NGV

The air distribution in gas turbine combustors is very much design specific and usually a secret within the industry. Hence a feasibility study done on this concept uses the data from an engine of 40KN thrust class turbofan civil engine design developed at Cranfield University as part of MSc thermal power group design project [Team1, 2009]. Engine parameters used are mentioned in the Table 3-1. The calculations are carried out for the take-off condition when the system is subjected to maximum load and mass flow. During take-off, the hot components in the gas turbine experience maximum temperature. The length and number of NGV blades are assumed to be 40mm and 25 respectively. This is selected on the basis of typical length and number of blades used in the engines of 40 KN thrust class civil turbo fan engines.

Table 3-1 Engine parameters at compressor and combustor

Engine parameters	**Values**
Compressor exit total pressure (P3)	2.8 MPa
FAR	0.22
Compressor exit total temperature (T3)	833K
Compressor exit mass flow (m3)	49.43kg
Turbine entry temperature (TET)	1540K
Compressor exit Mach number (M3)	0.562
Number of NGV Blades	25

The important step in validating the concept is to find out what would be the mass flow at take-off that would flow through each NGV blade when the tertiary region of the combustor is eliminated. The air distribution data after the compressor is very important to carry out this calculation.

For a take-off mass flow rate of 49.43 kg/s, cooling air flow rate (m_c) is assumed as 15.32kg/s, mass flow through dilution holes (m_d) is assumed as 8.9kg/s. The NGV cooling flow which would have passed through NGV blades with the tertiary region in the combustor is still expected to be passed through the same blades, since no change in the engine mass flow rate is assumed and elimination of NGV cooling flow mass from the total mass flow means complete design change through to the inlet fan. Hence the NGV cooling flow (m_{NGVc}) is assumed to be 2.96 kg/s. Hence the total flow through the NGV blades (m_{NGV}) is the summation of $m_{c,}$ $m_{d,}$ $m_{NGVc.}$

m_c + m_d+ $m_{NGVc.}$ = m_{NGV} (27.2kg/s.) Equation 3-1

To find the cross sectional area needed to pass the calculated amount of air, the following procedure is followed.

Form basic principles, Volume flow rate = Area × Velocity Equation 3-2

Mass flow rate = Mass/Time. Equation 3-3

Density (ρ) = Mass / Volume. = Mass flow rate / (Area × Velocity) Equation 3-4

Considering the total flow through NGV to be directly bled from the compressor exit, at a Mach number 0.562 and at T3 833K, Velocity/√Temperature can be found from the compressor tables.

From annex 3 of the compressor design manual [Team1, 2009]

$\frac{V}{\sqrt{T}} = 10.873$ Equation 3-5

We know that T = 833 K, hence V= 313.81m/s

Area = Mass flow rate/ (Density× Velocity) Equation 3-6

Considering the flow does not undergo any compression after the compressor exit, and that the flow is incompressible, the density at pressure can be found from ideal gas law.

$$\rho = \frac{P}{RT} = 11.71\ \text{kg/m}^3 \qquad \text{Equation 3-7}$$

Substituting value of ρ in to equation we get Area needed for the entire flow as $6.6\times10^{-3}\text{m}^2$

To check the suitability and feasibility of the proposed novel NGV, the NGV blade dimensions and shape is necessary. Cranfield University's MSc group design project of 2009 does not discuss anything about the NGVs.

This blade profile is selected based on the type of engine from which the previous data are obtained. The blade profile is similar to the NGV blades currently observed in modern aero-engines [Ramsden, 2010]. The area is calculated as $6.10 \times 10^{-4}\ \text{m}^2$. Simple arithmetic calculation lead to the conclusion that to accommodate all the amass flow through available 25 blades, 48.5% of the cross sectional area needs to be hollow in each blade. This means that such systems are physically possible.

The Author has tried a number of combinations/configurations for the dilution air inlet hole geometry for the NGV blade as shown below in Figure 3-9 and Figure 3-10.

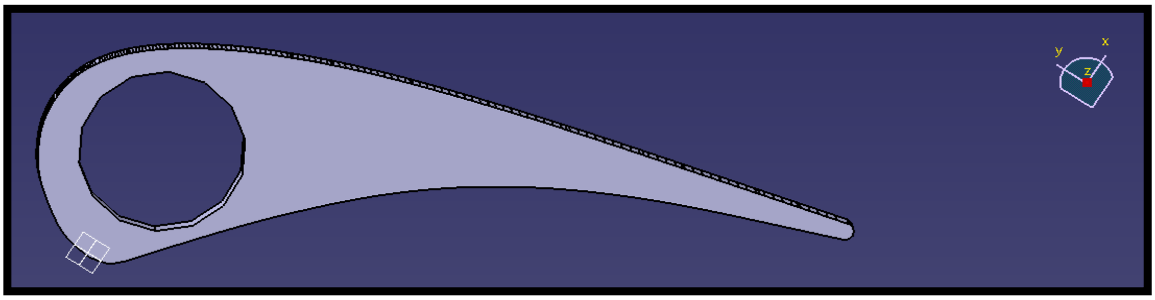


Figure 3-9 Dilution air inlet hole geometry configuration 1

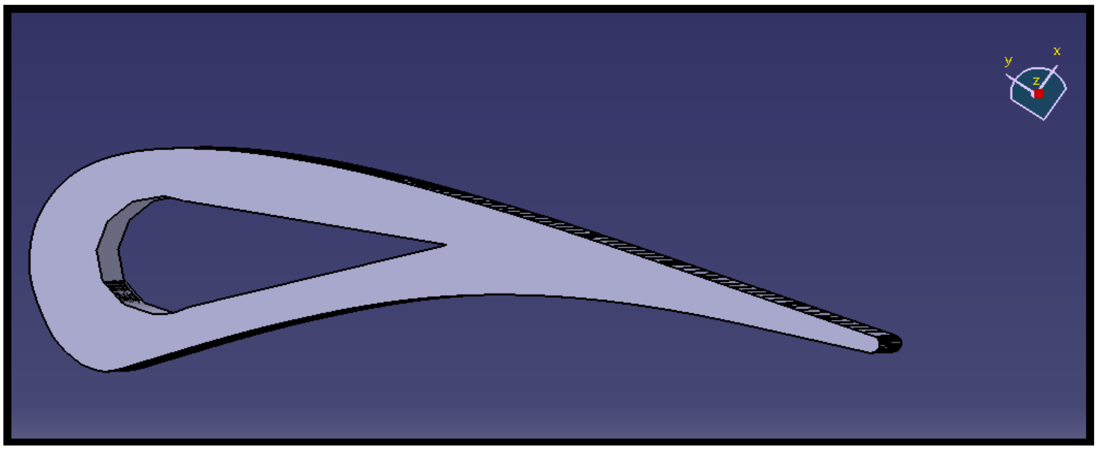


Figure 3-10 Dilution air inlet hole geometry configuration 2.

Since the idea is to input the dilution air from both ends of the NGV (Top and bottom end) the total air flow can be further divided into two equal parts. We assume that the air passing through both the ends are equal which in reality cannot be exactly predicted. Therefore the dilution air passing through one end of the conceptual NGV is 0.5 kg/s. Assuming number of holes on one side of NGV as 20 it is calculated that dilution hole diameter size is ~4 mm. By help of detailed discussions and basic calculation it is concluded that proposed configuration of NGV is feasible. From the study done it is found that the new concept is mechanically possible to construct without any major design changes in an already existing engine.

The concept although has advantages such as very good economy, weight saving and cut down in emission, the drawbacks too are worth a discussion. Channelling huge volume of air from the exit of the compressor in to the NGVs without considerable pressure losses is very difficult. If the pressure losses are more than the combustor pressure loss, then the hot efflux from the intermediate zone of the combustor would enter in to the NGV blades. it is also very important to analyse the changes in the expected life of the NGVs as they are subjected to higher thermal and hoop stresses. The design would result in a complex plumbing system for the air to be passed in to NGVs after the exit of the compressor. Hence any work towards the implementation of this concept needs to consider all the negative factors in to consideration and try to solve as many as possible.

3.5 Discussion on Patent Search

A detailed patent search has been done by author and patent office on this proposed concept of novel combustors. Several studies have been done on novel concepts of combustors. Different studies related to proposed novel concept have been discussed in subsections. The differences in concept proposed and earlier concepts have been mentioned in the subsections.

3.5.1 Gas turbine engine combustion equipment

Stanley Frank Smith from "Rolls Royce" proposed a concept [Smith, 1963] related to combustor parts as shown in Figure 3-11. The concept behind this invention is that the NGV (31) as shown in Figure 3-11 is integrated within the combustor chamber and cooling air/dilution air is injected into the flame tube region through the passages/holes present on the leading edge of the NGV. Here the cooling air that is injected through the holes (34) on the NGV is received from the dilution duct (27). This causes the cooling air to mix well with the combustion gases in the downstream direction lowering the combustion gas temperature prior to being introduced to the turbine blade (33).

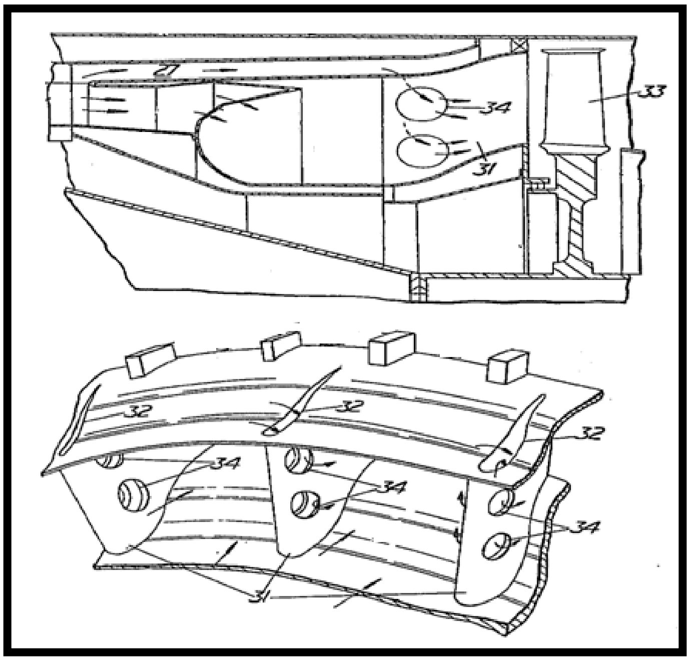


Figure 3-11 Radial section of the combustion chamber showing the invention [Smith, 1963]

By doing so the author claims to have cut down the length required for complete mixing of the gas stream and dilution air to half than that required by designs having holes on the single face of the NGV. High cooling mass flow is drawn in by this design which helps in ensuring better life span of the NGV. Finally, the author claims to have reduced the size of the combustor by inventing these concepts of improved mixing of the dilution air and main gas stream air and achieved lighter engine component hence reducing the overall weight of the engine.

The report submitted by the author does not mention whether the dilution zone of the combustor has been eliminated and whether the temperature profile at the exit of the NGV is uniformly maintained by this concept or not and the emission level obtained due to this design invention. The concept proposed by author is based on providing appropriate temperature traverse to turbine blade which achieving emissions goals.

3.5.2 Turbine Structure – Combustion structure

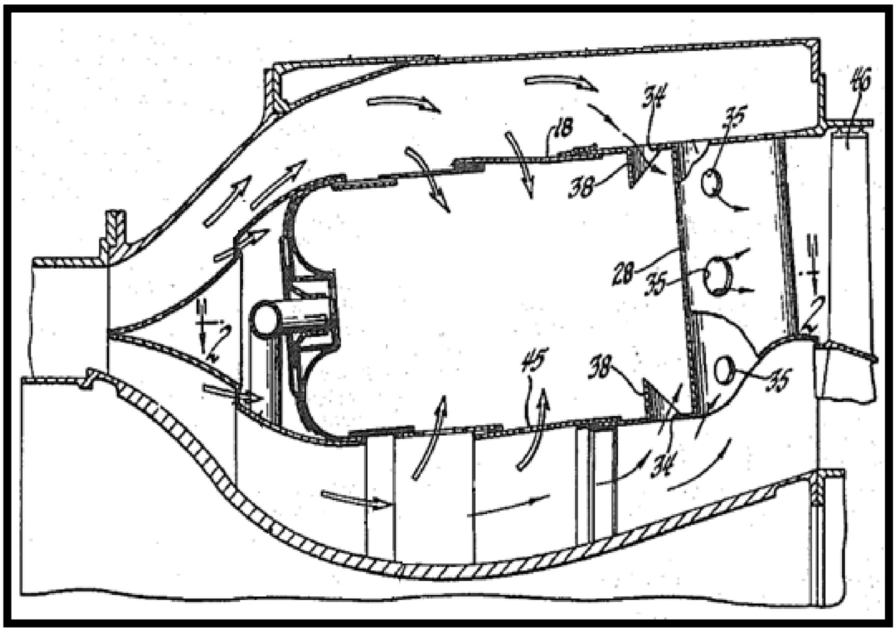


Figure 3-12, Integrated turbine stator-combustor in accordance to the invention [Vaught, 1966]

As per the patent filed by J.M. Vaught under the patent file number 3608310 given in the reference [Vaught, 1966], the invention relates to integrating the turbine stator (28) into the combustion chamber as shown in Figure 3-12. It is claimed by the author, that the new design concept has worked in favour of axially reducing the length of the gas turbine engine. This concept was introduced by the author for vertical lift engines such as the VTOL (Vertical Take Off and Landing) engines.

Also a concern raised by the author is that, as the distance of the stator with respect to the combustor primary zone is reduced, it lowers the temperature limit of the stator and hence more cooling air is required to be passed through the secondary combustor zone to protect the stator/NGV. Hence in-order to overcome this issue, the author brings about a unique “Secondary cooling air injecting” technique through protruding tubes (38 & 34) such as structure placed exactly in-front of the leading edge (36) of the stator vane/NGV on either ends of the liner of the combustor (18 & 45). Doing so ensured cooling air to impinge onto the leading edge (36) of the NGV, protecting it from the gas stream and also helped in diluting the main gas stream with the cooling air provided. Here the author aims to decrease the temperature of the combustion gas and provide impingement cooling to the leading edge (36) of the NGV. The major claims that is being stated is that by this invention the conventional combustion zone length to height ratio has seen a reduction from 2.5 to 1.5. The author also speaks about satisfactory temperature profile at the exit of the NGV and ultimately advantageous in reducing the length of the engine making the engine lighter.

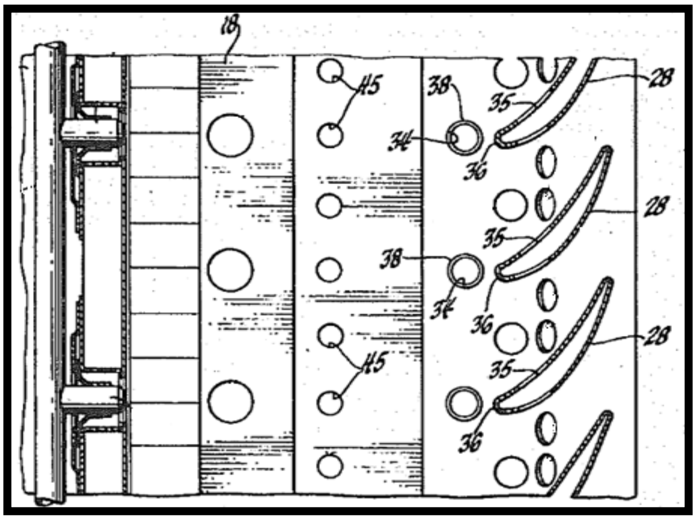


Figure 3-13 Sectional view of the integrated turbine stator-combustor [Vaught, 1966]

This report has no claims of having the dilution zone of the combustion chamber being eliminated and usage of NGV for passing the dilution air flow into the combustor. This report uses impingement cooling for leading edge of nozzle guide vanes. Also the report doesn't state whether there is appropriate temperature profile obtained at the exit of the NGV and the emission level obtained due to this design invention.

3.5.3 Cooled Vanes

As per the patent filed by Collin Godfrey & Rodney Carr Webster from Rolls Royce, under the patent file number 2189553 given in the reference [Collin and Webster, 1986], the invention relates to cooled vanes (19) suitable for the gas turbine engine and also this invention states to improve the internal cooling technology of the annular stator /NGV blade which are subjected to very high temperatures from the combustion gases from the combustor. Hence inorder to achieve the internal cooling of the NGV, cooling air is fed from the compressor section and being introduced into the internal passage of the NGV through the radially inner end (23) and radially outer end (24) of the NGV blade (19). The author states that the cooling air supplied through the radially inner end (23) is at higher pressure than the air supplied from the radially outer end (24) as shown in Figure 3-14 and Figure 3-15. The blade is also divided into two portions internally. One portion caters cooling air to leading edge (25) and the other caters cooling air to the side (30 & 31) and the trailing edge (20) of the blade. Both these portions are separated from each other (29) and hence do not influence each other in any form. Once the internal cooling of the blade is achieved, the cooling air is ejected out through holes (26) (39) & (42) on the leading edge, Suction side and pressure side of the blade and through the holes (41) provided on the trailing edge (20)of the NGV (19). This helps in providing a film cooling air around the blade which is used for protecting the blade from the hot gas stream.

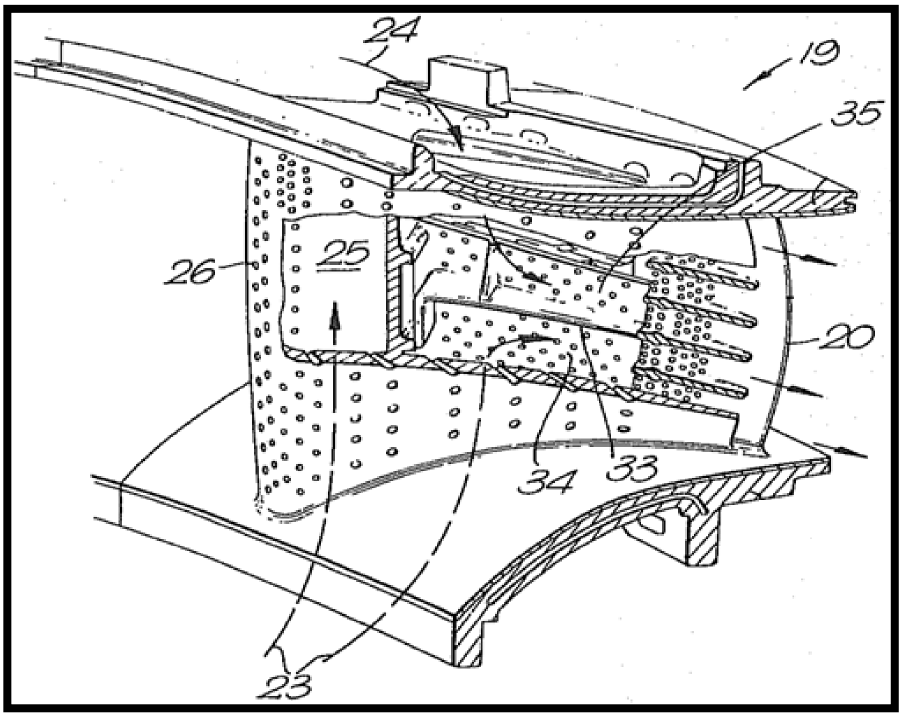


Figure 3-14 Partial broken away perspective view of the cooled vane invention [Collin and Webster, 1986]

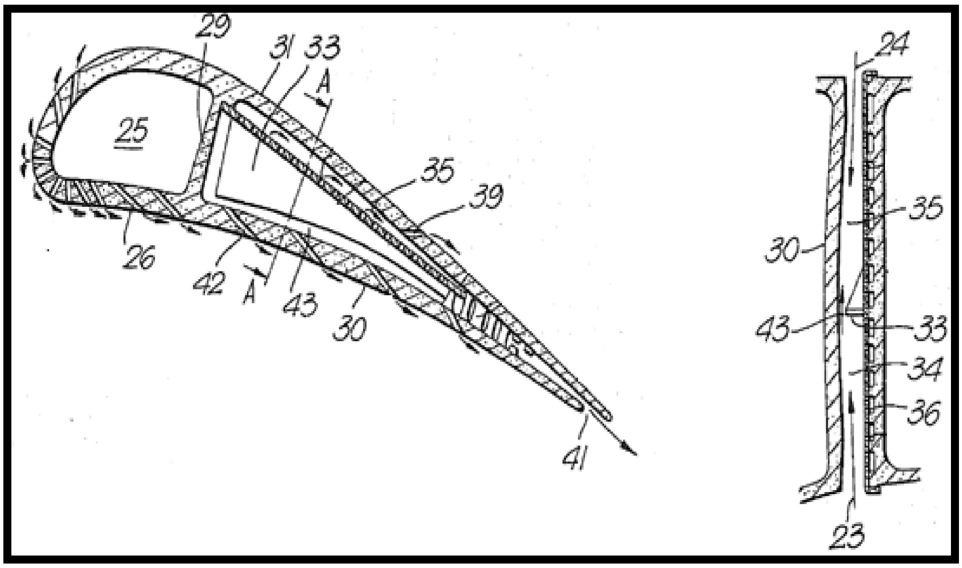


Figure 3-15, Sectional view in radial direction of the aerofoil section of the vane [Collin and Webster, 1986]

This report has no claims of having the dilution zone of the combustion chamber being eliminated and usage of NGV for passing the dilution air flow into the combustor nor does it speak about appropriate temperature profile obtained at the exit of the NGV by mixing the cooling air to drop the combustion air temperature, nor does he speak about the emission control achieved by this design. Here the author mainly concentrates on the internal and external cooling of the NGV.

3.5.4 Dilution Pole combustor and method:

As per the patent filed by Richard W. Sticles, Loveland et al. from General Electric Company, under the patent file number 5239818 given in the reference [Stickles et al, 1993], the invention relates to inserting a cooling pole (14) in the intermediate zone (20a) of the combustor as shown in Figure 3-16 and injecting cooling air (22a) through the holes (48) provided on each side of the dilution pole for dropping down the main gas stream temperature and for efficient mixing of both. Here the author states that a series of dilution poles (14) have been inserted between the forward combustion zone and after end of combustion zone, such that the alternative poles are bolted down to either end of the liner (16 & 18). By inserting the poles (14) the author tends to reduce the passage area for the main gas steam flow and hence accelerates the gas flow which could reduce the residence time and ultimately cause a reduction in NOx. Also claims have been made that the poles (14) are made hollow through which the cooling air (22a) could be injected into the passage (44) between the alternative dilution poles where the main gas stream (38) accelerates, which would improve the chances of mixing of the combustion gases (38) with the cooling air (22a) and ultimately cause an reduction in temperature which is less life threatening to the NGV (32).

This invention is helpful in improving the axially staged, rich-lean combustor operation as claimed by the author. Also the rich combustion gas is made to mix with the cooling air coming from the dilution poles, which helps in converting it into a lean mixture, later which under goes complete combustion at the after-zone of the combustion chamber and along with reduced residence time would reduce NOx formation.

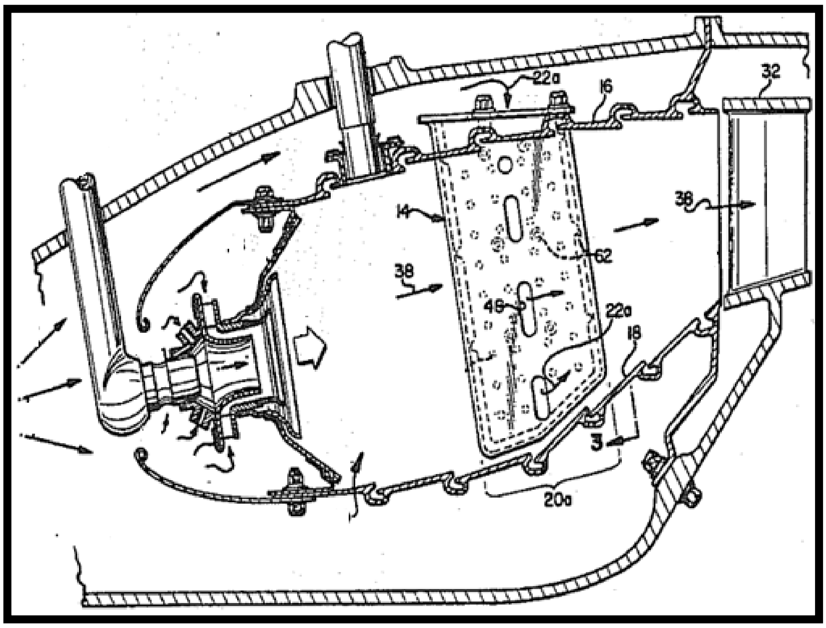


Figure 3-16, Axial, partly sectional view of one half of combustor including dilution pole [Stickles, 1993]

This report has no claims of having the dilution zone of the combustion chamber being eliminated and usage of NGV for passing the dilution air flow into the combustor nor does it speak about uniform temperature profile obtained at the exit of the NGV.

3.6 Advantages

There are many advantages that can be derived from the application of this concept. The proposed idea reduces the combustor length without altering the temperature profile requirements for the rotor. The main advantages here are,

- Possibility of reduction in the residence time of combustion products in the hottest region, resulting in reduction in NOx formation.
- This design allows developing a shorter combustor thereby reducing the overall weight of the engine and dropping fuel consumption.
- Possibility of engine being more economic and less polluting.
- Shorter length of the combustor chamber gives rise to possibility of reduction in cooling air requirement due to reduced area.
- Higher mass flow through NGV blades would eliminates the need for complex internal machining usually done for cooling purposes. This is

because, as the velocity and mass flow increase, convection between the NGV surface and air also increases.

- Reduced length and cooling requirements have positive effects on the combustor material selection and manufacturing contributing to simplicity and economy in general.
- Possibility of maintain uniform temperature profile at the exit of the NGV since the NGV is used to dilute the hot gas stream
- Possibility of reduction in cooling air requirement in the combustor due to the elimination of tertiary zone thereby could allow additional air to take part in the main thermodynamic process.

Reduced length and cooling requirements have positive effects on the combustor material selection and manufacturing contributing to simplicity and economy in general.

3.7 Conclusions

A preliminary study has been done for the concept is to eliminate the tertiary zone of an annular combustor and divert all the mass flow through to the NGV's that would have otherwise passed through dilution holes of tertiary region. It is found that such systems with huge flow rates can be practically implemented without any major design changes in the engine. The calculations are being carried out for the NGV dilution holes at the leading edge. Various NGV entry hole geometry also being formed through which the entire tertiary zone mass flow could be passed. Discussion from patent search has yielded that proposed new concept is a new idea. An experimental study is required on the proposed novel combustor for substantiating this concept. The proposed novel dilution zone less combustor has led to substantial contribution to knowledge in field of combustors.

4 NOVEL COMBUSTOR WITH HYDROGEN SYNTHESIS FROM KEROSENE REFORMATION

4.1 Abstract

Addition of hydrogen as an additive in gas turbine combustor shows large benefit with respect to both performance and reduction in NO_x levels. Due to complexity associated with dual fuel injection, it has not been implemented in gas turbines. In this study a novel combustor concept is proposed whereby a hydrogen rich mixture is synthesised within the combustor, thereby enhancing the combustion process. It is perceived that the novel combustor has two stages. Combustion of ~5% of the hydrocarbon fuel occurs in the first stage at a high equivalence ratio (above 1.5) in the presence of a catalyst, consequently producing hydrogen rich flue gases. In the subsequent stage hydrogen rich flue gases from first stage act as an additive to combustion of hydrocarbon fuel. Preliminary studies on the proposed combustor are presented in this paper. The preliminary studies demonstrate that the mixture of hydrocarbon fuel and air could be burned at equivalence ratios as low as 0.4–0.5, giving better temperature profiles and larger stability limits than conventional kerosene fuelled combustors. Computational and equilibrium analysis demonstrate reductions in NO_x, CO and CO_2 with increased hydrogen synthesis levels. Lower casing temperature due to less emissivity of hydrogen flame and combustion at lower equivalence ratios leads to increase in life of combustor. This method of combustion has led to significant contribution to knowledge in field of combustors.

4.2 Introduction

Gas turbines are compact, lightweight, easy to start with reliability and simplicity to operate. It has been widely used for many years in industries like power generation, marine and aero propulsion systems. In recent years the emission reduction for conventional aero gas turbines burning fossil fuels, especially in civil aviation has attracted interests of many researchers. Generally, conventional aero gas turbines exhaust gases contain carbon dioxide (CO_2), water vapour (H_2O), oxides of nitrogen (NO_x), unburned hydrocarbons (UHC), carbon monoxide (CO) and oxides of sulphur (SO_x) as main combustion products. CO_2, H_2O, NO_x, SO_x and particles are of most concern with respect to climate perturbations [Rogers, 2002]. At ground level NO_x emission results in increase in ozone concentration, prolonged exposure to ozone may cause respiratory illness, impaired vision, headaches and allergies [Lefebvre, 2010]. Moreover, NO_x emissions cause damage to plant life as well as add to the problem of acid rain. At high altitude (above about 15 km) NO_x emissions causes ozone depletion leading to increased ground-level ultraviolet radiation, which might cause skin cancer and eye disease [Singh, 2011]. Table 4-1 shows the effects of several pollutants from gas turbine. With the growth of population and per capita demand in air traffic is expected to increase at a rate of ~5 %, the sustainability of fossil oil resources raises concerns. How to control pollutant emissions has become a great issue due to their harmful influence on people's health and the environment. Hence, both regulations for pollutants produced by gas turbines and the technologies used to control emissions to meet these regulations have experienced rapid changes in the last several decades since the gas turbines became one of the most important power generators both in civil aviation and industrial application. With the above concerns, study on low emission combustors, especially low NO_x, becomes favourable in the combustor design for aero gas turbines.

Table 4-1 Pollutants and their effects

POLLUTANTS	**EFFECTS**
Carbon monoxide (CO)	Toxic, Ozone depletion
Unburned hydrocarbon (UHC)	Toxic
Oxides of nitrogen (NO_x)	Toxic, precursor of chemical smog, depletion of ozone in stratosphere
Particulate matter (C)	Visible
Oxides of sulphur (SO_x)	Toxic, corrosive

Researchers around the world have studied and proposed several methods to reduce emissions and include sustainable alternative methods to run gas turbines. Some of them include powering aircraft by electricity [Gohardani, 2011], bio fuels [Simon, 2011; Rye, 2012], synthetic fuels [Simon, 2011; Rye, 2012], and hydrogen [Sevensson, 2004; Higland, 2006; Hart, 1999; Khandelwal, 2011 (c)]. Different configurations of gas turbine combustors have also been proposed by companies and researchers for decreasing the emissions including staged combustor, lean premixed pre-vaporised combustor, catalytic combustor, EV combustor, water/steam injection method [Lefebvre, 2010] and micro-mix combustor [Khandelwal, 2011 (c)].

One of the methods to reduce emissions is by using hydrogen as a fuel, due to its various advantages over other alternatives. Specific energy of hydrogen is 2.8 times higher than liquid hydrocarbon fuel. From combustion viewpoint hydrogen has high flame speed, flame stability limits and low emissivity which lead to minimizing metal temperature and thermal stresses [Jackson, 2009]. Hydrogen combustion could be sustained for a wide range of hydrogen-air mixtures (4% to 75%) [Boehman, 2008]. Due to excellent heat transfer properties and specific heat capacity as a liquid, it is highly effective as a heat sink to be used as coolant.

On the other hand there are some disadvantages of using hydrogen as a fuel in aero gas turbines. Hydrogen has very low density and boiling point which necessitates the large, heavily insulated storage tanks on the aircraft.

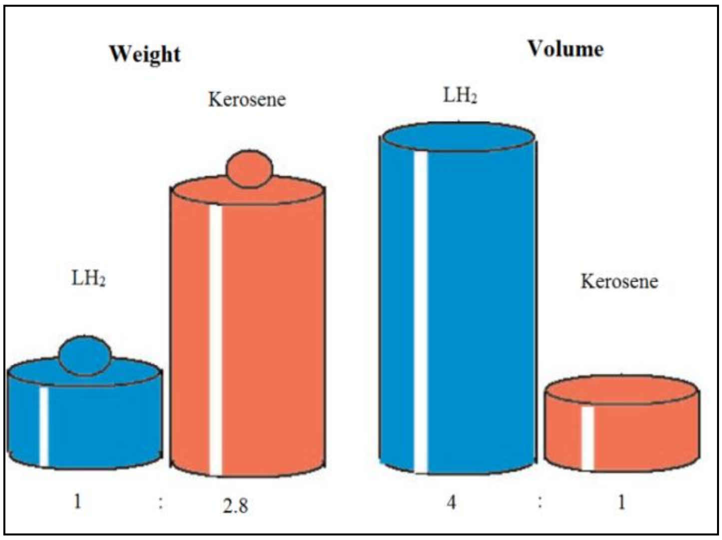


Figure 4-1 Comparison of LH_2 and Kerosene Regarding Weight and Volume [Nojoumi, 2009]

Table 4-2 Properties of Hydrogen, Kerosene and Natural Gas

	Units	Hydrogen	Kerosene (Avtur)	Typical Natural Gas
Lower Heating Value	MJ/Kg	120.24	43.2	48.0
Density – Liquid	Kg/Litre @15 ^{0}C	0.07(At Boiling Point)	0.8	N/A
Energy Density Of Liquid, Net	MJ/ Litre	8.42	34.6	N/A
Density Of Gas, Ntp	Kg/M^3	0.085	N/A	0.74
Specific Heat Cp, Asa Gas, 288k	J/Kg K	14310	N/A	2130
Gas Constant, R	J/Kg K	4124.5	N/A	475.5
Specific Heat Ratio, Y At 288k	--	1.4	N/A	1.287
Boiling Point	^{0}C	-252.7	150-260	N/A

Hydrogen is a highly flammable compound; flashback is the major problem which can cause explosion before hydrogen air mixture enters into combustor. Most important if there is any leakages in storage it could reach to troposphere and destroy the methane – scrubbing OH radicals, and could also reach the stratosphere leading to the halogen, destruction of ozone [Jackson, 2009]. Table 4-2 shows different properties of Hydrogen, Kerosene and Natural gas.

In recent years substantial research have been done by different researchers across the globe on introducing hydrogen as an additive fuel to conventional gas turbine combustors [Gobbato, 2011; Juste, 2006; Frenillot, 2009; Burguburu, 2011] and IC engine [Green, 2000; Boehman, 2008]. Addition of hydrogen not only reduces emission it also improves combustion properties of the combustors. It has been found that combustion of hydrocarbon fuel enriched with hydrogen in small quantity, leads to substantial increase in combustion performance. For reducing NO_x emissions in combustion of hydrocarbon fuel, lean combustion is proposed as an alternate, but it comes at a cost of lower efficiency and increased CO and UHC emission. Researchers have found that hydrogen injection enhances the stable domain of combustion towards leaner equivalence ratios [Gobbato, 2011; Juste, 2006; Frenillot, 2009].

A recent experimental study by Juste [2006] on effect of hydrogen injection as an additional fuel in gas turbine shows that hydrogen injection reduces emissions substantially. Juste found that by injecting up-to 4 % hydrogen to lean primary zone leads to decrease in emission index of CO by 30 %. It was also found that addition of small quantities of hydrogen leads to substantial reduction in CO_2 emissions if same energy input is maintained to the combustion chamber. Burguburu et al. [2011] experimentally studied a spray injection system for aviation kerosene to investigate the effect of hydrogen enrichment at constant power for its use in airplane engine. Two injection configurations were tested in the study, one was based on partially premixed combustion, whereas, other was based on premixed combustion as shown in Figure 4-2. At high temperatures and pressures it is found that at both injection systems can operate with higher hydrogen inlet mass flow rate i.e. energy contribution (EC) of 7.4 %, without much effect of global flame shape. Lean blowout limit reduced from 0.53 to 0.35 for most of the favourable conditions. It is also found that partially premixed combustor configuration as shown in Figure 4-2 gives wider flame stability limits as compared to premixed configuration. A similar phenomenon is also observed in micro-mix combustors [Khandelwal, 2011 (c)]. CO emissions were reduced by a factor of 4 by small quantity addition of hydrogen due to enhancement of reaction by hydroxyl radicals.

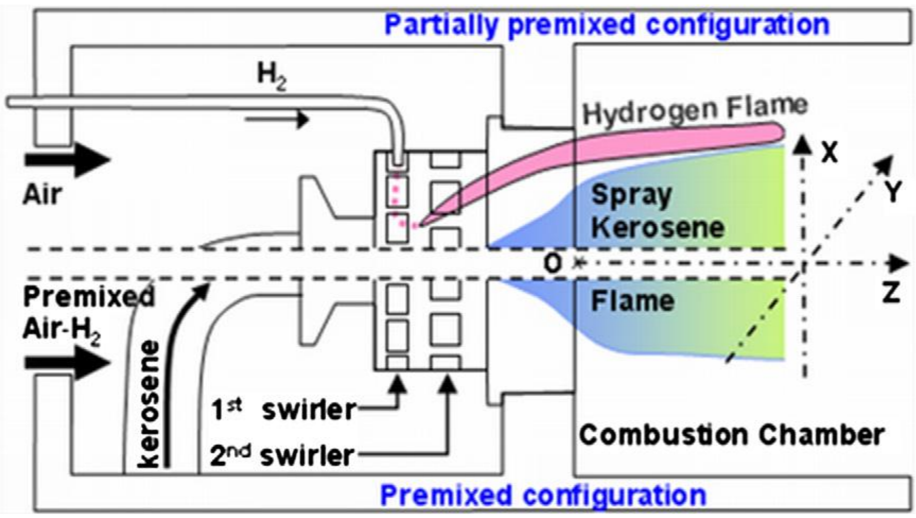


Figure 4-2 Schematic explanation of flame stabilisation [Burguburu, 2011]

Frenillot et al. [Frenillot, 2009] studied effect of hydrogen addition in a lean gas turbine combustor on flame stability limits and pollutant emissions. It is observed that with hydrogen addition, flame remains stable at much leaner equivalence ratios as compared to flame only with kerosene. In the case of 10% of hydrogen injection, the flame at 0.4 equivalence ratio is still anchored to the nozzle and stable, whereas, flame extinguished in the case of pure kerosene/air mixture combustion. 25 % gain in combustion flame stability is observed with hydrogen addition up-to 10 %. Figure 4-3 shows variation in NO_x emissions with varying percentage of hydrogen addition at different equivalence ratios. A computational and experimental study by Gobbato [2011] has shown that hydrogen enrichment improves the temperature profile both in axial and radial direction. This leads to optimum temperature traverse for turbine inlet.

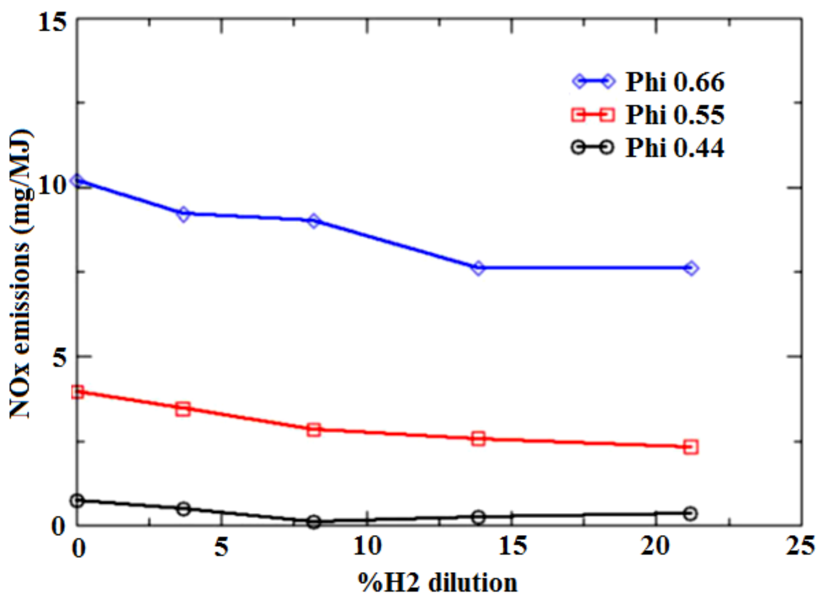


Figure 4-3 NO_x emissions for constant combustion power [Frenillot, 2009]

4.3 Methodology

Several studies have shown that there are substantial benefits of enriching kerosene combustion with hydrogen for improvement in combustion performance. This result in enhanced flame stability limit, better temperature profile, increased altitude relight capability and lower emissions. Due to the complexity associated with dual fuel injection, it has not been implemented in gas turbines. Aircraft engine operating on this principle would have to carry two fuels which increase the complexity, manufacturing, storage and maintenance problems. In this study a method has been proposed by which the complexity of carrying two fuels on-board will be eliminated. In this study a novel combustor concept is proposed whereby a hydrogen rich mixture is synthesised within the combustor, thereby enhancing the combustion process. It is perceived that the novel combustor has two stages. Combustion of ~5% of the hydrocarbon fuel occurs in the first stage at a high equivalence ratio (above 1.5) in the presence of a catalyst, consequently producing hydrogen rich flue gases. In the subsequent stage the hydrogen rich flue gases from the first stage act as an additive to combustion of the hydrocarbon fuel. In this concept it is proposed that first stage combustor would be significantly small then the second stage combustor. The first stage combustion could even be performed in an engine or aircraft as it requires small space. Flue gases would be eventually fed into the second stage combustion chamber where majority of combustion is occurring, leading to improved performance.

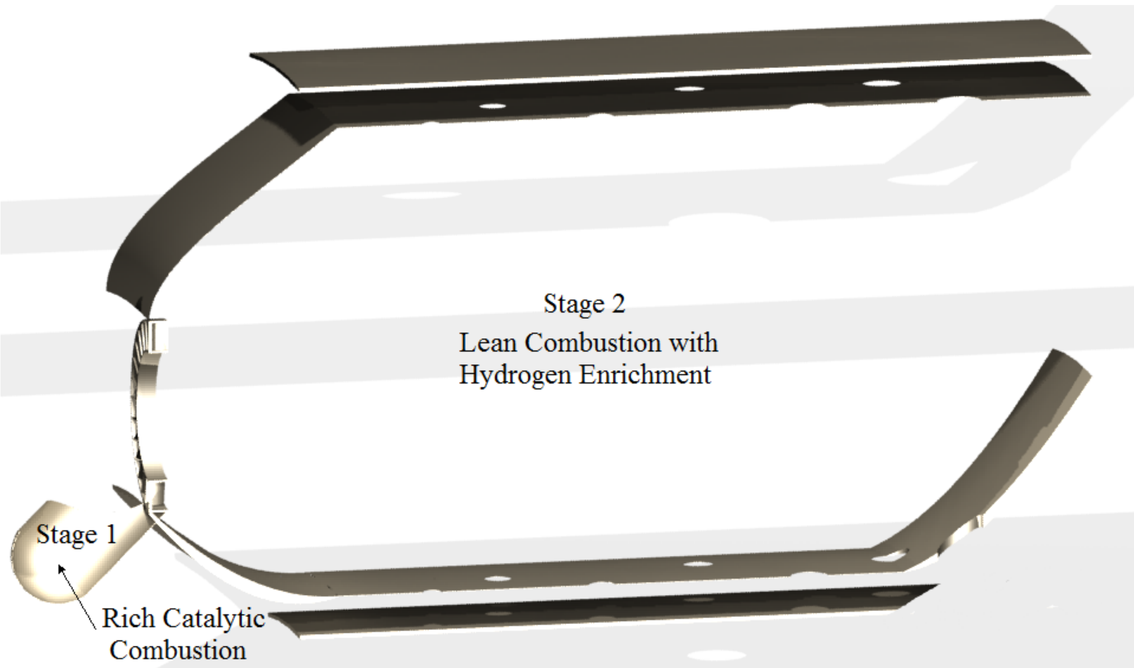


Figure 4-4 Proposed novel two staged combustor [Khandelwal, 2012 (b)]

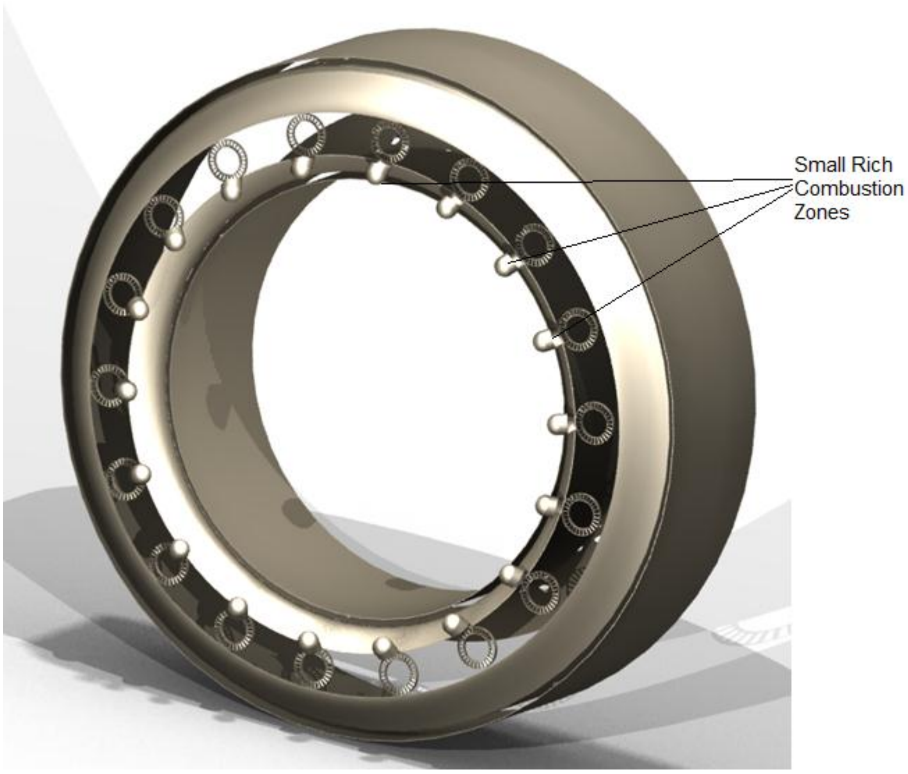


Figure 4-5 Proposed arrangement of small rich combustion zones [Khandelwal, 2012 (b)]

A detailed research has been carried out on hydrogen production and enrichment of kerosene combustion. An equilibrium study has also been done to predict the performance of rich combustion for hydrogen production. Computational method has been used to check the performance of hydrocarbon fuel combustion with varying hydrogen percentage. There are several configurations for this concept of combustor which could be devised. Figure 4-4 shows an indicative schematic of proposed combustor. Figure 4-5 shows arrangement of rich combustion zone in an annular combustor. In the following section various methods to produce hydrogen enriched gas on board have been discussed.

4.4 Hydrogen Production methods

There are many methods by which hydrogen could be produced; some of them are electrolysis of water, rich combustion of hydrocarbon fuel with catalyst and reforming of fossil fuels. Presently, majority of hydrogen gas is produced from fossil fuel, including coal, natural gas, refinery tail gases and coke over gas, as this is most economical method known till date. Hydrogen derived from fossil sources is done using reforming of natural gas according to mechanism shown in Eqn. 1. Partial oxidation of methane is done to produce hydrogen as shown in Eqn. 2. Electrolysis of water is not substantially used due to inefficient and expensive method of production for hydrogen.

$$(CH_4 + H_2O = CO + 3H_2) \quad \text{....(1)}$$
$$(2CH_4 + O_2 = 2CO + 4H_2) \quad \text{....(2)}$$

Commercial production of hydrogen is not done by rich combustion of hydrocarbon fuel in presence of catalyst due to its significant inefficiency and emissions of unburned hydrocarbon and CO. Whereas, this method could be used to produce hydrogen rich gases for gas turbine combustor, as these gases would be further combusted for burning unburned hydrocarbon fuel. Several studies have been done by researchers for producing hydrogen by rich combustion of hydrocarbon fuel in presence of catalyst or porous media. Producing hydrogen by rich combustion of hydrocarbon fuel and supplying it for conventional lean combustion seems to be an alternate for increasing combustion properties of hydrocarbon fuel.

Bingue et al. [2004; 2002] studied production of hydrogen by filtration combustion of methane. Bingue et al. [2004; 2002] carried out an experiment on 45 cm long and 3 mm diameter tube with a packed bed of Al_2O_3used for burning methane at various equivalence ratios. 25 % of total products are converted into hydrogen at an equivalence ratio of 2.75 and oxygen content of oxidiser as 35 %. In another study by Cheekatamarla and Finnerty [2008] on catalytic partial oxidation reforming of liquid fuels, it was found that significant amount of hydrogen and CO could be produced by the process.

Loukou [2012] had carried out an experiment on Stationary Thermal Partial Oxidation (TPOX) with inert porous media to study Hydrogen production and soot emission. In order to examine operating characteristics, he had carried out this experiment in two different porous materials namely SiSiC [Silicon infiltrated silicon carbide] and Al_2O_3 (Aluminium Oxide). For operation regime of inlet temperature of 400^0C to 550^0C, thermal load varying from 350 kW/m^2 to 2600 kW/m^2 and equivalence ratio from 1.9 to 2.9, he found that SiSiC metrics due to high porosity shows higher thermal properties and allows longer residence time for slow reforming reaction, toward equilibrium [Loukou, 2012]. It was found that for a range of higher equivalence ratio and higher operating temperatures with preheating, soot is not observed.

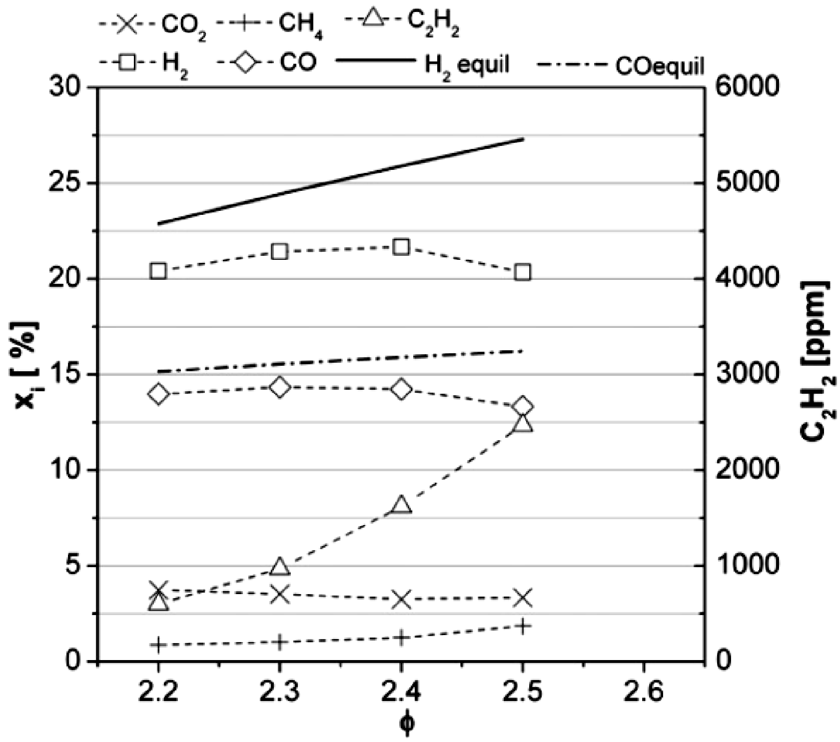


Figure 4-6 Exhaust gas composition [Loukou, 2012]

Figure 5 shows exhaust gas comparison of minor and major species for experimental study on hydrogen production from methane rich combustion in inert porous media. Al_2O_3 have been used as a catalyst with preheated air at 400° C. Equilibrium concentrations of H_2 and CO have been plotted with experimental results for comparison.

Similar study by Pederson-Mjaanes et al. [2005] examines the concept of hydrogen production by rich combustion of octane, methane, methanol and automotive-grade petrol in inert porous media. Pederson-Mjaanes found that around 56% of methanol is converted into Syngas (H_2 and CO) inside alumina foam burner whereas 66% yield of syngas in alumina bead burner. Whereas, conversion from methane and octane to syngas leads to lower efficiency of less than 45%. It is also found that methanol has highest equivalence ratio for stable combustion in comparison to methane, octane and petrol. It is also proposed that its quick start-up time, high turn-down ratio and compact size would make it potential candidate for future fuel cell powered automobile. Syngas consist of hydrogen and CO which could be used as enrichment in novel gas turbine combustor proposed in this study. Toledo et al. [2009] in their study about hydrogen production in ultra-rich combustion of hydrocarbon fuels in porous media found that maximum hydrogen yield is close to 50 % for all the fuels considered in the study. In another study on conversion of emulsified kerosene in a gas generator with catalytic reforming it was found that hydrogen content volume of 4~16 % can be achieved in fuel rich combustion [Hou, 2010]. Hamelinck [2001] had conducted a study on evaluating future production of hydrogen and methanol from biomass both technically and economically. It is concluded that energy efficiency is around 55% HHV for methanol and around 60% for hydrogen production.

Several other studies had been done by other researchers on hydrogen production by steam reforming [An, 2003; Suzuki, 2000; Hou; 2010], and catalytic reforming [Lomax, 2002; Matzakos, 2004; Lutz, 2003; Wild, 2000], using catalyst like Ru/Al_2O_3 [Suzuki, 2000]. Numerous studies mentioned above shows that there is a potential to produce hydrogen by rich combustion in presence of catalyst to be used in gas turbine combustor. It is also to be noted that volume required for production of hydrogen enriched gases is small which could be eventually fitted in a gas turbine combustor without major changes. This hydrogen enrichment in a gas turbine combustor will increase the flame stability limits substantially. Combustion could be done at much lower equivalence ratios as compared to conventional cases without hydrogen enrichment.

4.4.1 Hydrogen Production without catalyst

In this study production of hydrogen by ultra-rich combustion of hydrocarbon fuel has been studied. Combustion with help of catalyst/porous media has not been considered in this study due to availability of several other studies in public domain. Equilibrium analysis technique has been used to calculate the products of ultra-rich combustion at various equivalence ratios and inlet temperatures.

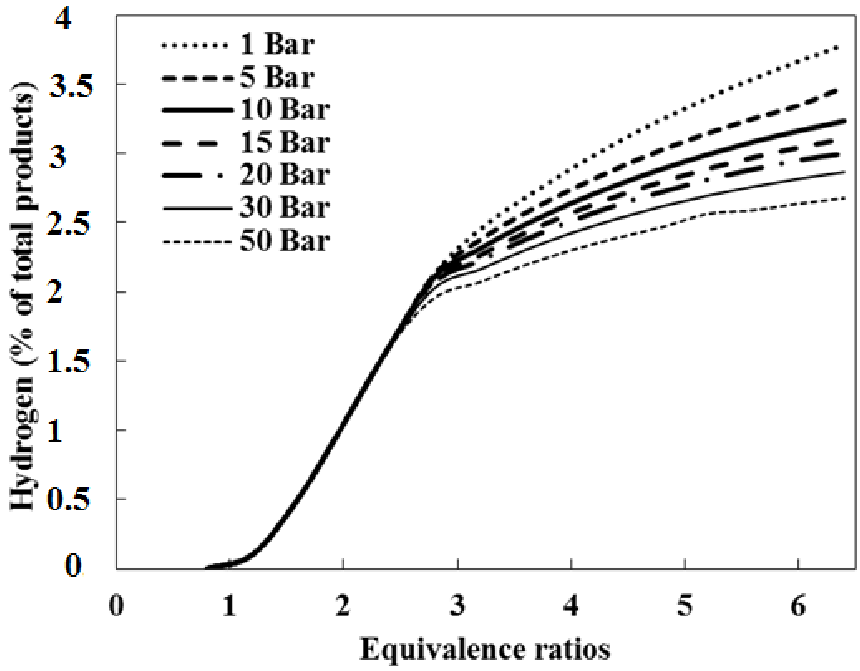


Figure 4-7 Hydrogen production percentage at different equivalence ratios [Khandelwal, 2012 (b)]

Figure 6 shows hydrogen produced at different equivalence ratios and combustion pressures at an inlet air temperature of 873 K. Hydrogen productions shown in Fig. 6 is in percentage of mass of total products. It is observed that at lower combustion chamber pressures hydrogen production is higher. 2.5 % - 4 % hydrogen is produced at above $\Phi > 3.0$. Similar observations are observed for air inlet temperatures of 973 K and 1073 K.

4.4.2 Effect of hydrogen enrichment on hydrocarbon combustion

In this work a range of preliminary computational analyses have been done to demonstrate the effect of hydrogen enrichment on hydrocarbon fuel combustion. Methane has been used as a main fuel and hydrogen has been added in various percentages to check the effect on combustion performance. The overall equivalence ratio has been kept same in combustion of methane and methane-hydrogen combustion. A micro-mix combustor [Khandelwal, 2011(c); Murthy, 2011] concept geometry have been selected to check the performance. Five different geometries have been studied to check the performance of combustion named as Case 1, Case 2, Case 3, Case 4 and Case 5. Based on the performance of different geometries, one is selected for further study with hydrogen injection in hydrocarbon combustion.

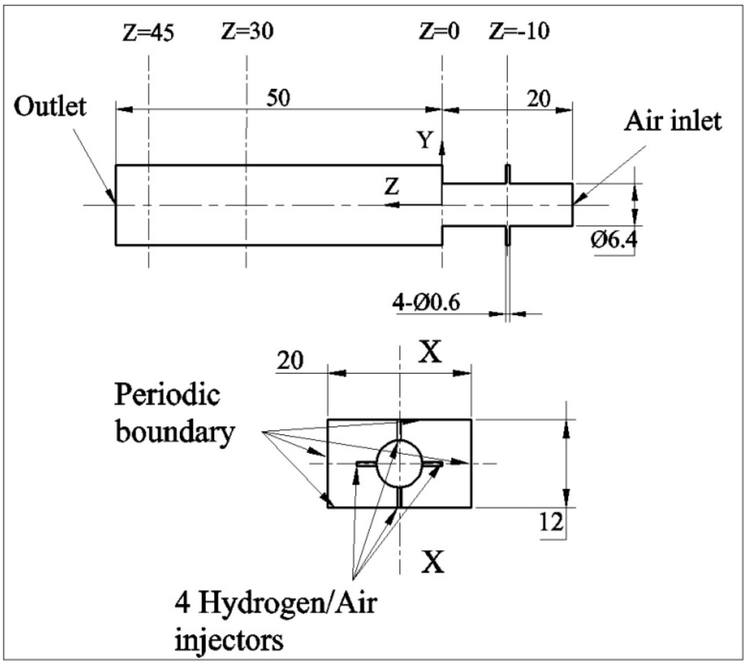


Figure 4-8 Micro-mix Combustor (Case 1) [Khandelwal, 2011(c)]

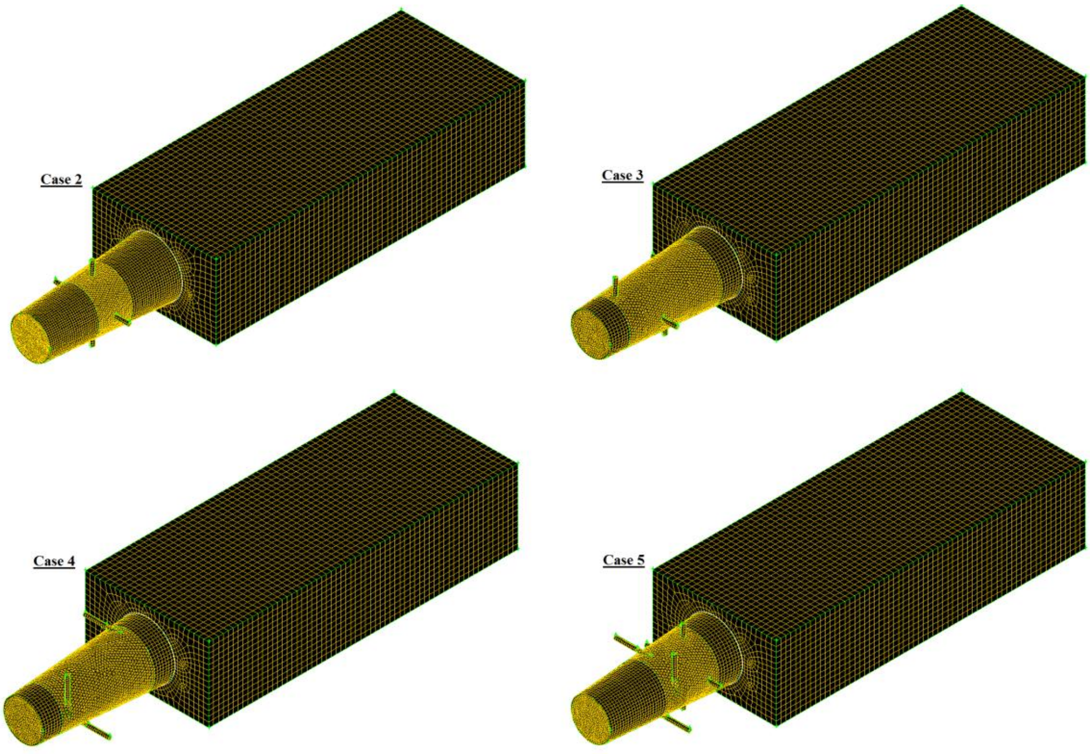


Figure 4-9. Illustration for case 2, case 3, case 4 and case 5.

Case 1 is based on the original micro-mix method used by Khandelwal et al. [2011 (c)] that will enable a good base line to compare additional cases. Case 2 injection section has been changed to diverging from cylindrical as in case 5, keeping 4 hydrogen/air injectors same. It is expected to improve the stability of the flame profile with improved mixing. In case 3 position of hydrogen/air injection have been varied. It will show how the injections positions will either increase flame length of reduce it; there is no dramatic improvement in mixing expected with this case. In case 4 hydrogen/air injections has been made tangential. It will enable the effect of a radial velocity to aid in anchoring the flame profile in much shorter distances which is expected to reduce emissions. In case 5 numbers of injectors for hydrogen/air has been increased as shown in Figure 4-10. It is expected to have the most beneficial mixing profile with the lowest emissions. The geometry will need to provide a balanced profile to impart a stable flame profile.

Geometric details of the case 5 combustion zone studied in this study have been shown in the Figure 4-10. A cuboidal shape combustion zone has been considered in this study to check the effect of hydrogen enrichment on combustion of methane. Mixture of air and fuel is being injected at z = -5 mm towards positive z direction for combustion. The boundary conditions of the combustion zone have been considered as periodic. FLUENT is setup to use a non-pre-mixed combustion model with a realizable k-ε viscosity model examining thermal NO_x.

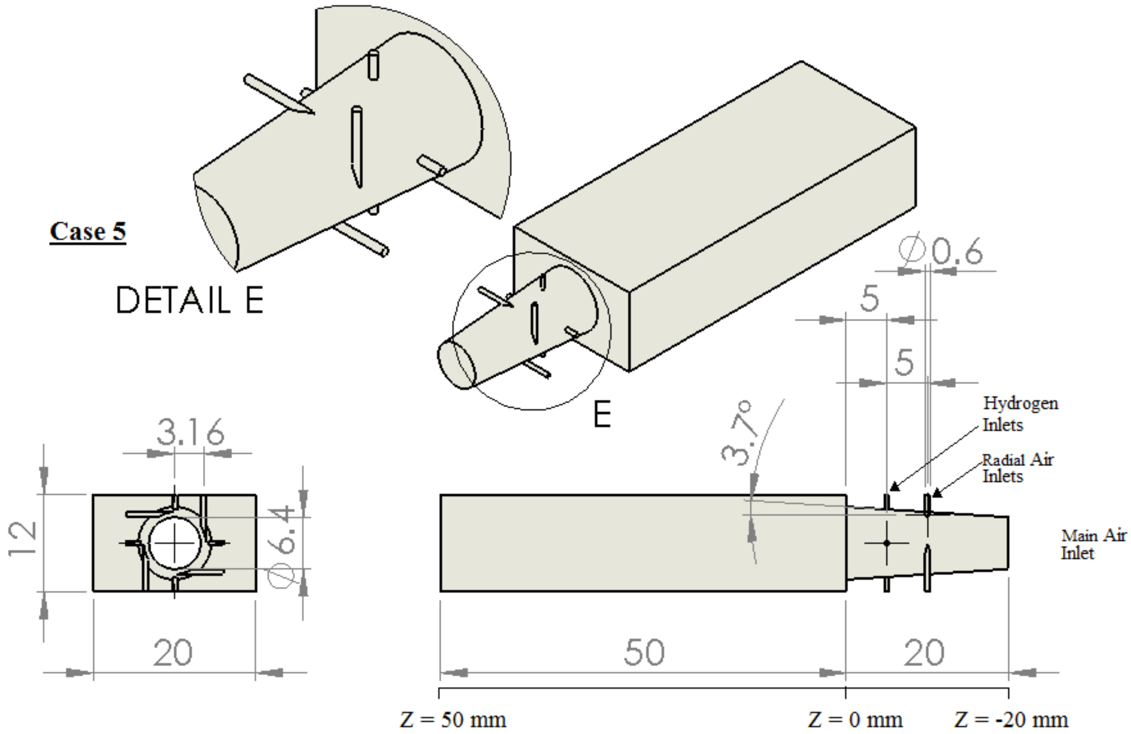


Figure 4-10 Geometry details

The CFD code used to simulate the above designs are validated by using it for Lean Direct Injection (LDI) hydrogen combustors, in which the data from the "N1 injector with 2.5 in liner" are selected [Marek, 2005]. The combustor exit temperature with corresponding equivalence ratios under a certain operation condition is selected as the reference data for the validation. All the numerical models and the corresponding settings for the validation study are kept coincident with those for the three combustors intended to be investigated. The operating conditions for the validation, corresponding to the selected test conditions, are listed in Table 4-3. The domain geometries for the validation investigation are shown in Figure 4-11.

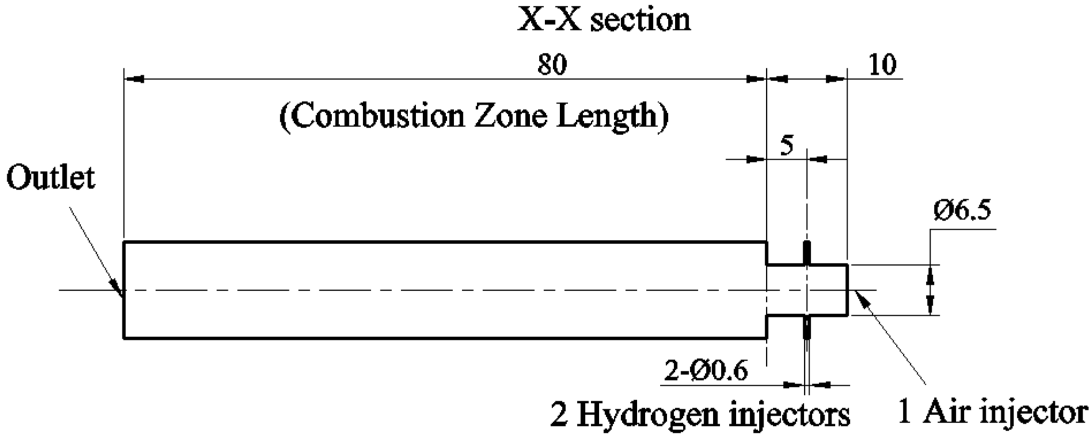


Figure 4-11 Domain geometries for the validation (mm) [Khandelwal, 2011 (c)]

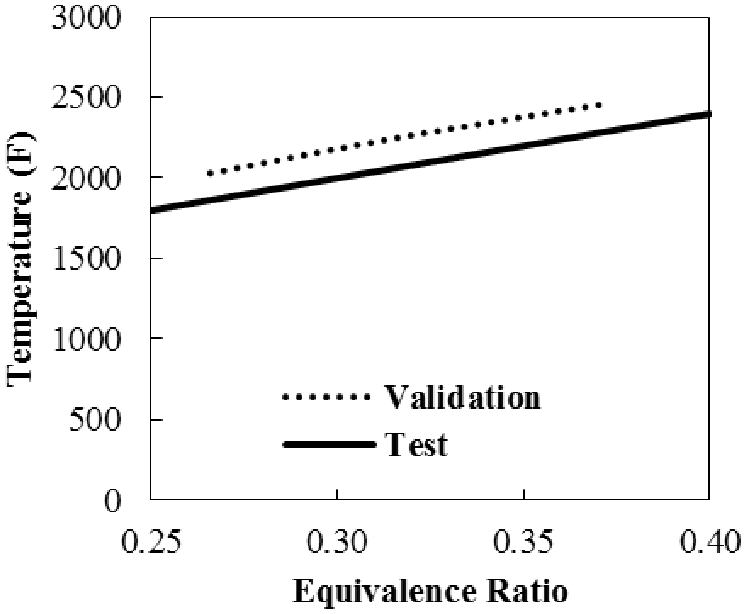


Figure 4-12 Comparison of the outlet temperature against the equivalence ratio between the validation data and the test data [Khandelwal, 2011 (c)]

The comparison of the outlet temperature against the equivalence ratio between the validation data and the test data is illustrated in Figure 4-12. The outlet temperatures, T_{out}, both in test data and validation are presented in the units of Fahrenheit. From this figure, small discrepancies can be found between the outlet temperatures obtained from the validation and from test under the same equivalence ratio. In addition, test data also have 10% variation on the equivalence ratio. Therefore, that small discrepancy found between test data and validation data are acceptable. To sum up, the above validation study indicates that the numerical models used in this case have a reasonable accuracy.

Table 4-3 Operating conditions for the validation

Parameters	Settings
Air inlet velocity (m/s)	30.5
Equivalence ratio	0.266, 0.319, 0.372
Operation pressure (MPa)	0.690
Inlet temperature (K)	600

It was found that case 5 gives better temperature profile and lower NO_x emissions. Case 5 have been used for further study by injection hydrogen in combustion of methane to check the effect of hydrogen addition to hydrocarbon combustion.

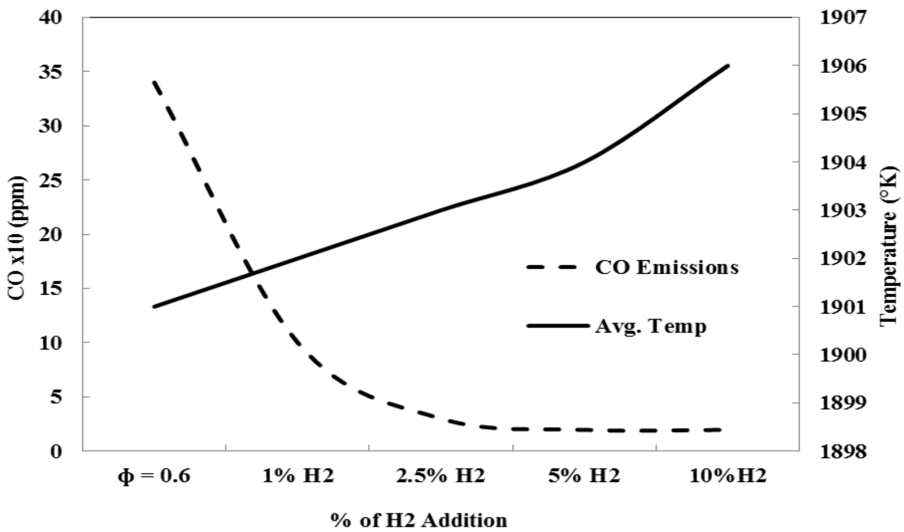


Figure 4-13 Effect of Hydrogen Addition to Methane combustion [Khandelwal, 2012 (b)]

Previous studies have shown that hydrogen addition to hydrocarbon combustion leads to decrease in emissions of CO and slight increase in temperature. The impact of increase in temperature can be could be reduced by burning the mixture on leaner equivalence ratios. Hydrogen addition to hydrocarbon fuel combustion helps in burning the mixture at much leaner equivalence ratios. It is also to be noted that the average temperatures do not increase by a significant amount, therfore the thermal NO_x content should not be altered with the reduction of CO emissions. Figure 4-13 shows the trend of CO emissions and average temperature across the exit cross section at ER = 0.6. The investigation of hydrogen addition has proven the trend of reduced CO emissions with H_2 addition as shown in Figure 4-13. With substantial emsision reductions aquired from 1-2.5% H_2 addition as previously shown by Juste [Juste, 2006].

4.5 Summary

Researchers around the world have studied and proposed benefits of including hydrogen as an additive in the hydrocarbon combustion. This would lead to increase in flame stability limits which would eventually aid in improving altitude relight performance. This preliminary study has provided an addition step towards the concept of using hydrogen as an additive by showing the effect of hydrogen addition on hydrocarbon fuel combustion. It has been observed that hydrogen addition of 2.5 % would lead to decrease in CO emissions from 35 ppm to 5 ppm. There is a substantial decrease in CO2 and NOx emissions. Addition of hydrogen as an additive in gas turbine combustor would increase complexity including production, transport, fuel tank and other supply chain difficulties. Production of hydrogen by kerosene reforming and then using hydrogen rich gases to aid combustion is a lucrative alternate. By extensive study of literature it has been found that ~50 % mass of fuel air mixture could be converted into hydrogen by aid of catalytic reforming of kerosene. Size of volume required for production of hydrogen rich gases is small; eventually this reaction zone could be easily accommodated within combustor or any other part of the aircraft. The proposed novel combustor would have two stages, combustion of ~5% of the hydrocarbon fuel would occur in the first stage at higher equivalence ratios in the presence of a catalyst, which would eventually lead to the formation of hydrogen rich flue gases. In the subsequent stage the hydrogen rich flue gases from the first stage would act as an additive to combustion of the hydrocarbon fuel. In this concept it is proposed that first stage combustor would be significantly small then the second stage combustor. Flue gases would be eventually fed into the second stage combustion chamber where majority of combustion is occurring, leading to improved performance. Ability to burn at leaner equivalence ratios leads to reduction in flame temperature which would eventually lead to decrease in casing temperature. This would lead to increase in life of combustor.

4.6 Conclusions

Preliminary study on the novel combustor shows that proposed novel combustor is feasible. Hydrogen rich products produced by ultra-rich catalytic combustion would improve performance of gas turbine combustor. Hydrogen addition leads to increase in flame stability limits. Problem of carrying two different fuels and their supply line for hydrogen enrichment would be eliminated by mounting the proposed combustor configuration. 20-40 % combustion products of ultra-rich combustion could be converted to hydrogen, which would eventually lead to enhancement of combustion performance. Experimental testing of the proposed novel combustor is required as a next step towards implementing this combustor in gas turbines. This concept has resulted in publication and eventually led to contribution to knowledge.

5 CONCLUSIONS & RECOMMENDATIONS

5.1 CONCLUSIONS

Design methodologies for different combustors have been developed based on empirical approach. The combustors for which design methodologies have been developed are single annular combustor, double annular radial and axial combustor, triple annular radial and axial combustor and reverse flow combustor. Combustor design methodology includes detailed designing of different parts of combustors which includes diffuser, fuel injector, swirler, casing, liner, cooling holes and other relevant parts. Design methodologies developed are resulting in combustors of reasonable resemblance with present combustors. Estimated performance analysis also shows that the combustors designed are giving appropriate performance. Developed design methodologies substantially add contribution to knowledge as there are no such design methodologies available in public domain.

The concept of hybrid diffuser has been around for a while and has the potential of giving a greater performance than conventional diffusers. However, due to limited studies available in the public domain, not much has been understood about the mechanism of the hybrid diffuser concept. Majority of the previous work done on hybrid diffusers is done on designs having a vortex chamber bleed, based on the belief that vortex chambers helps to stabilize the flow separation. However, this study takes looks into the proposition that the primary mechanism of a hybrid diffuser is the air bleed rather than the vortex chamber itself. This study looks at a preliminary study between a dump diffuser, hybrid diffuser with a vortex chamber and that of a new hybrid diffuser design where the vortex chamber is replaced with a duct bleed. Different performance parameters have been analysed at different bleed rates. The results show that the duct bleed hybrid diffuser has a similar performance to that of a vortex chamber hybrid diffuser. However, it was observed that a duct bleed needed even less bleed air to achieve a good performance. Performance analyses on different configurations of duct bleed hybrid diffusers have been presented in this study. These configurations include changes in post-diffuser divergence angle, bleed out angle, profile bleed, inside bleed and changes in the dump gap. Changes in diffuser angles do give a better performance but at the expense of larger amount of air bleed from the mainstream. Reducing the bleed angle has shown no substantial gain to the diffuser performance but improved the quality of bleed air which can be used for other purposed like turbine cooling. Inside bleed configuration of hybrid diffuser is not giving standard performance.

A preliminary study has been done for the concept is to eliminate the tertiary zone of an annular combustor and divert all the mass flow through to the NGV's that would have otherwise passed through dilution holes of tertiary region. It is found that such systems with huge flow rates can be practically implemented without any major design changes in the engine. The calculations are being carried out for the NGV dilution holes at the leading edge. Various NGV entry hole geometry also being formed through which the entire tertiary zone mass flow could be passed. Discussion from patent search has yielded that proposed new concept is a new idea. An experimental study is required on the proposed novel combustor for substantiating this concept. The proposed novel dilution zone less combustor has led to substantial contribution to knowledge in field of combustors.

Preliminary study on the novel combustor shows that proposed novel combustor based on hydrogen synthesis by kerosene reforming is feasible. Hydrogen rich products produced by ultra-rich catalytic combustion would improve performance of gas turbine combustor. Hydrogen addition leads to increase in flame stability limits. Problem of carrying two different fuels and their supply line for hydrogen enrichment would be eliminated by mounting the proposed combustor configuration. 20-40 % combustion products of ultra-rich combustion could be converted to hydrogen, which would eventually lead to enhancement of combustion performance. Experimental testing of the proposed novel combustor is required as a next step towards implementing this combustor in gas turbines. This concept has resulted in publication and eventually led to contribution to knowledge.

5.2 RECOMMENDATIONS

Preliminary step by step design methodologies have been developed for various combustors. An interactive program needs to be developed by help of an appropriate coding language for giving out an appropriate tool for preliminary combustor designing. Detailed emissions analysis methodologies need to be inherited for accurate prediction of emissions.

In this study preliminary analyses have been performed for various novel combustor concepts. For the proposed combustor with hydrogen synthesis by kerosene reforming experimental analysis for verifying the performance is required. Various parameters need to be investigated including the effects on flame stability limits and emissions of injecting hydrogen in a gas turbine combustor. Energy input to the combustor needs to be same as that of hydrocarbon fuel based combustor. Further study should also emphasise on overall cost effectiveness and adaptability of the conceptual combustor for commercial aviation. Accurate details of increase in altitude relight capability have to be determined. The concept also needs to be checked with a range of catalysts and without catalyst.

Dilution zone-less combustor is predicted to be a feasible idea. Detailed computation analysis of the proposed combustor is required to check the effect of various parameters including hole size on NGV, angle of hole, quantity of air to be injected from NGV, effect on temperature traverse and effect on life of NGV. After detailed computation analysis, verification of the parameters is required by help of experimental testing. Detailed analysis on benefits of shorter length combustor on weight, length, air flow distribution and fuel consumption will be required before checking the overall cost effectiveness of the concept.

APPENDICES

Appendix A explains different preliminary design methodologies developed. Design methodologies include double annular combustor, triple annular combustor and reverse flow combustor. Appendix B explains different novel vortex controlled hybrid diffusers examined. Appendix C enlists patents, publication with their abstracts and awards received during the PhD study.

Appendix A Combustor Design

A.1 Triple Annular Combustor

A.1.1 Abstract

Modern gas turbine combustor design is a complex task which includes both experimental and empirical knowledge. Numerous parameters have to be considered for combustor designs which include combustor size, combustion efficiency, emissions and so on. Several empirical correlations and experienced approaches have been developed and summarized in literature for designing conventional combustors. A large number of advanced technologies have been successfully employed to reduce emissions significantly in the last few decades. There is no literature in the public domain for providing detailed design methodologies of triple annular combustors.

The objective of this study is to provide a detailed method designing a triple annular dry low emission industrial combustor and evaluate its performance, based on the operating conditions of an industrial engine. The design methodology employs semi-empirical and empirical models for designing different components of gas turbine combustors. Meanwhile, advanced DLE methods such as lean fuel combustion, premixed methods, staged combustion, triple annular, multi-passage diffusers, double cooling walls, DACRS and heat shields are employed to cut down emissions. The design process is shown step by step for design and performance evaluation of the combustor.

The performance of this combustor is predicted, it shows that NO_x emissions could be reduced by 60%-90% as compared with conventional single annular combustors.

A.1.2 Introduction

A gas turbine combustor is an essential part of the engine which consists of a diffuser, swirler, fuel injector, primary zone, dilution zone as well as cooling systems [Lefebvre, 2010]. Many design methodologies have been developed for designing conventional aircraft engine combustors. However Mellor [Mellor, 1990] indicated that most of the design rules for conventional aero combustors can also be applied to other combustor applications and configurations. For aircraft combustors size and weight are more important factors, whereas for industrial combustors more attention is given to its resilience and durability. Reduced emissions and fuel consumptions are desired for both of the combustor [Lefebvre, 2010; Mellor, 1990; Singh, 2011] types. Proven aircraft combustor design methods can be used for the preliminary design of industrial combustors. Most industrial combustors are derived from a successful aero-combustor. The design methodology employs semi-empirical and empirical models for designing different components of gas turbine combustors.

Mohammad and Jeng [Mohammad, 2009] have proposed a design procedure for the preliminary design of a single annular combustor and its performance. In one of the studies recently published by our group members [Khandelwal, 2011 (a)], the design procedure of a reverse flow combustor for a helicopter engine with high temperature rise has been presented. Lefebvre [Lefebvre, 2010] and Mellor [Mellor, 1990] collected an extensive list of empirical and analytical tools. The objective of this study is to provide a detailed method designing a triple annular dry low emission industrial combustor and evaluate its performance, based on the operating conditions of an industrial engine. In this research paper an approach to designing an industrial combustor has been discussed by applying conventional design methods and modern low emission technology. The combustor performance has also been predicted after calculating the dimensions of main structure.

Table A-1 Design Point Performance of 2010 GDP Engine

OPR	TET (K)	Airflow (kg/s)	Pressure of Nozzle (atm)	Shaft Power (MW)	Efficiency (%)	SFC (kg/MW.s)
27.46	1529.97	53.69	1.054	18.57	40.96	0.05677

The operating conditions of an industrial gas turbine have been taken from an engine which is designed by the 2010 Group Design Project (GDP) [Group1, 2010] of the Thermal Power Course in Cranfield University. In the group design project an existing aero engine was modified into an industrial engine operating at sea level, providing around 19MW with a thermal efficiency of around 41%. The engine model has been established and simulated by using TURBOMATCH (Cranfield University's in-house gas turbine performance evaluation code). Hence the results obtained from this simulation have been regarded as the basis of this study. Results from the GDP are the foundation of this study owing to the fact that substantial literature is not available in the public domain.

Table A-2 Off-Design Point Performance of 2010 GDP Engine

Ambient Temp (°C)	TET (K)	OPR	OP (MW)	Fuel flow (kg/s)	SFC (kg/MW.s)	Heat Rate (kJ/kWh)	Eff (%)
-40	1530	35.943	26.98	1.4732	0.05460	8256	42.47
-30	1500	33.833	24.24	1.3333	0.055	8317	42.16
-15	1530	31.793	22.77	1.2638	0.05549	8392	41.79
0	1550	29.86	21.13	1.1862	0.05611	8484	41.32
15	1580	28.26	19.97	1.1314	0.05667	8566	40.92
15	1530	27.463	18.57	1.0542	0.05677	8583	40.85
30	1600	26.842	18.67	1.0708	0.05735	8672	40.44
40	1600	25.641	17.39	1.0098	0.05807	8780	39.93

Table A-1 shows the design point performance of the GDP engine. Table A-2 shows its performance when the ambient temperature varies from -40^{o}c to 40^{o}c. To meet the dry low emission requirements, Double Annular Counter Rotating Swirlers (DACRS) and staged combustion concepts have been adopted in this industrial combustor. NO_x emissions can be reduced by enlarging the combustor volume whilst increasing the number of fuel injectors with the growth in volume. CO and UHC emissions can be maintained to a low level due to the longer residence times [Lefebvre, 2010].

Natural gas has been considered as a fuel for this engine because of its industrial applications. Advanced low emissions technology has been adopted in this design, using staged combustion, lean premix pre-vaporize concepts and double wall cooling considerations.

A.1.3 Design Procedure and Results

Operating condition for initiating design procedure

The input parameters for the design of an industrial gas turbine combustor have been acquired from the GDP Report of the Thermal Power Course in Cranfield University. The main input parameters of the combustor are shown in Table A-3. The total length, inlet and outlet diameters have been taken as 0.335m, 0.495m and 0.563m respectively from the GDP engine design. For an industrial gas turbine engine it is necessary to use gaseous fuels due to their low price, reduced emissions and mixing properties.

Table A-3 Main Input Parameter of the Combustor

Inlet Temperature (K)	TET (K)	Airflow (kg/s)	Inlet Total Pressure (kPa)	Inlet Velocity (m/s)	Liquid Fuel Flow (kg/s)	SFC (kg/MW.s)
801	1530	48.53	2746	130	1.054	0.05677

Fuel consumption of liquid fuel in the GDP engine is 1.054 kg/s . Consumption of natural gas could be calculated by using the following calculation process, according to Pilidis [Pilidis, 2010]:

$$W_{in}*Cp_{in}*T_{in}+W_{ff}*\mathrm{Cp}_{\mathrm{gas}}*T_{gas}+\mathrm{LHV}*W_{ff}*\ ETA=(W_{in}+W_{ff})*Cp_{gasou}t*T_{out} \qquad (1)$$

Where:

W_{in} (inlet air flow of combustor) = 48.53 kg/s;

$\mathrm{Cp}_{\mathrm{in}}$ (specific heat of inlet air) =1000 kJ/kg°K;

T_{in} (inlet temperature of combustor) =801°K;

Cp_{gas} (specific heat of gaseous fuel) =1050 kJ/kg°K;

T_{gas} (temperature of gaseous fuel) =801°K;

LHV (Low Heat Value of gaseous) =48.2 MJ/kg;

ETA (efficiency of combustion) =0.999;

Cp_{gasout} (specific heat of outlet combustion gas) =1080 kJ/kg°K;

T_{out} (temperature of outlet gas) =1530°K.

W_{ff} (fuel flow) (kg/s);

W_{ff} is calculated as 0.859 kg/s.

For most types of lean fuel combustion, the equivalence ratio (ϕ) is always in ranges from 0.4 to 0.7. An equivalence ratio of 0.6 is selected for lean combustion in this study. The actual fuel to air ratio in the primary zone can be calculated from the equivalence ratio. The ideal fuel to air ratio for methane as a gaseous fuel is 0.058. Hence, the actual fuel to air ratio (F.A.R.) can be obtained from following equation:

$$\text{F.A.R.actual} = \text{Equivalence Ratio}\,(\phi) * \text{F.A.R.ideal} = 0.6*0.058 = 0.0348$$

The amount of air required for complete combustion can be calculated using the following expression:

$$\text{Combustion Air Flow Rate} = \frac{\text{Fuel Flow Rate}}{\text{F.A.R.actual}} \quad (2)$$

Combustion air flow rate is calculated as 24.68 kg/s.

Considering that a portion of the dome cooling air and liner cooling air will take part in combustion, the actual combustion fuel to air ratio may be lower than that calculated above. It implies that the actual equivalence ratio is lower than 0.6. However, it does not influence the flame stability as only a small portion of the cooling air will take part in the reaction. Conversely, lower equivalence ratios will result in a larger reduction of NO_x emissions. The operating conditions of an industrial combustor under study are shown in Table A-4.

Table A-4 Operating Conditions of Industrial Combustor

Name	Value(DP)
Inlet air flow (kg/s)	48.53
Inlet total temperature (K)	801
Inlet total pressure (kPa)	2746
Outlet total temperature (K)	1530
Fuel flow (kg/s)	0.859(natural gas)
Equivalent ratio	0.6
F.A.R ideal for gaseous fuel	0.058(natural gas)
F.A.R	0.0348
Air flow for combustion (kg/s)	24.68

In this section an industrial combustor will be designed step by step using the empirical and semi-empirical methods, whilst considering industrial requirements. Mellor [Mellor, 1990] has provided a list of empirical and semi-empirical tools for designing conventional combustors. In addition B. S. Mohammad [Mohammad, 2009] has introduced his preliminary design procedure of a conventional single annular combustor.

Generally the design of a combustor [Mohammad, 2009; Khandelwal, 2011 (a)] is examined from inlet to outlet. This permits the inlet diffuser, combustion chamber flow-path, combustor primary zone, dilution zone and liner cooling methods to be designed individually. This study will concentrate on the design of each component with its own regulations and design criteria. While considering the whole combustion system design requirements including operations, performance, configurations, emissions and durability.

Diffuser Design

There are several types of diffusers used in modern gas turbine combustors. A multi passage diffuser is selected in this design task because this type of diffuser gives low pressure losses within a relatively short length and large divergence angles. In this design, one or more annular splitters are employed to form multi passages through the combustor dome region where high energy air flows. The length of the diffuser can be reduced by this method to obtain a large diffuser area ratio. The air flow is then divided by the splitter vanes into two or more high area ratio passages without flow separation. Cochran and Kline [Cochran, 1958] in 1958 had demonstrated this advanced concept could work successfully. Their research showed that low pressure losses and un-stalled air flow can be obtained with total divergence angles up to 42^{o}.

There are also some successful applications of splitter vanes. One is the NASA/GE “Energy Efficient Engine” which adopts a single splitter vane to meet the large area ratio requirement of 1.8 and divides the air flow into two passages directly into outer and inner annuli with nearly the same mass flow. The length of this two passage diffuser is reduced by 50% relative to a single annular diffuser with the same area ratio. Another successful multi passage diffuser is the GE LM6000 diffuser. The primary zone volume is almost twice that of a conventional combustor in order to reduce its emissions. A four passage diffuser is used to provide air flow to the larger volume without increasing the diffuser length or total combustor length. It is for this reason that a four passage diffuser is employed in the industrial combustor of this study as its primary zone volume is large in this design.

Firstly, the inlet area and height of diffuser can be calculated. The static temperature (t_3), static pressure (p_3), air density (ρ), and air flow velocity (V_3) at the inlet of diffuser can be acquired by Eq. (6). Inlet airflow Mach number has been taken as 0.23.

$$\mathrm{t}_3 = \frac{T_3}{\left(1+(\gamma-1)/2M_3{}^2\right)} \tag{3}$$

$$p_3 = (t_3/T_3)^{\gamma/(\gamma-1)} * P_3 \tag{4}$$

$$\rho_3 = \frac{p_3}{R * t_3} \tag{5}$$

$$V_3 = M_3 * \sqrt{\gamma * R * t_3} \tag{6}$$

Results obtained for t_3, p_3, ρ_3 and V_3 are 792.6°K, 2646.7kPa, 11.63kg/m^3 and 129.8m/s respectively. Using this data the inlet area (A_{di}) and height (H_{di}) of the diffuser can be calculated.

$$A_{di} = \frac{m_3}{\rho_3 * V_3} \tag{7}$$

$$H_{di} = \frac{A_{di}}{2 * \pi * r_3} \tag{8}$$

The inlet radius can be found from the GDP engine which is 0.24m. The inlet radius, height, inlet outer radius and inlet inner radius are 0.247m, 0.021m, 0.258m and 0.237m respectively for the diffuser. Taking the LM6000 engine's diffuser as a reference, whose area ratio is about 2-2.5 and length to height ratio is around 3-4. The area ratio of this diffuser design is selected to be 2.3 and the length to height ratio as 3.5. Hence the outlet area of the diffuser is around 0.074m^2 and the length of diffuser is about 0.07m. The average angle of the diffuser is selected as 8^o. Therefore the outlet section dimensions are calculated to be 0.257m for radius, 0.045m for height, 0.281m for outer radius and 0.233m for inner radius.

The divergence angle of each passage and the overall divergence angle need to be calculated. Based on the inlet and outlet dimensions, the overall divergence angle is calculated as 32^o and each passage has a divergence angle of 8^o. Consequently the inlet and the outlet radius of each passage could be calculated keeping the area of each passage the same. The schematic of the designed diffuser has been shown in Figure A-1. After the dimensions of the diffuser are calculated, the total diffuser pressure losses can be estimated. The pressure recovery coefficient, C_p, can be calculated form the following equation considering the area ratio of the diffuser as 2.3 and the length to height ratio as 3.5.

$$C_p = -0.918*((LW)^{-1} * AR^{-0.5}) + 0.677*(LW^{-2} * AR^{-2}) + 0.74 \tag{9}$$

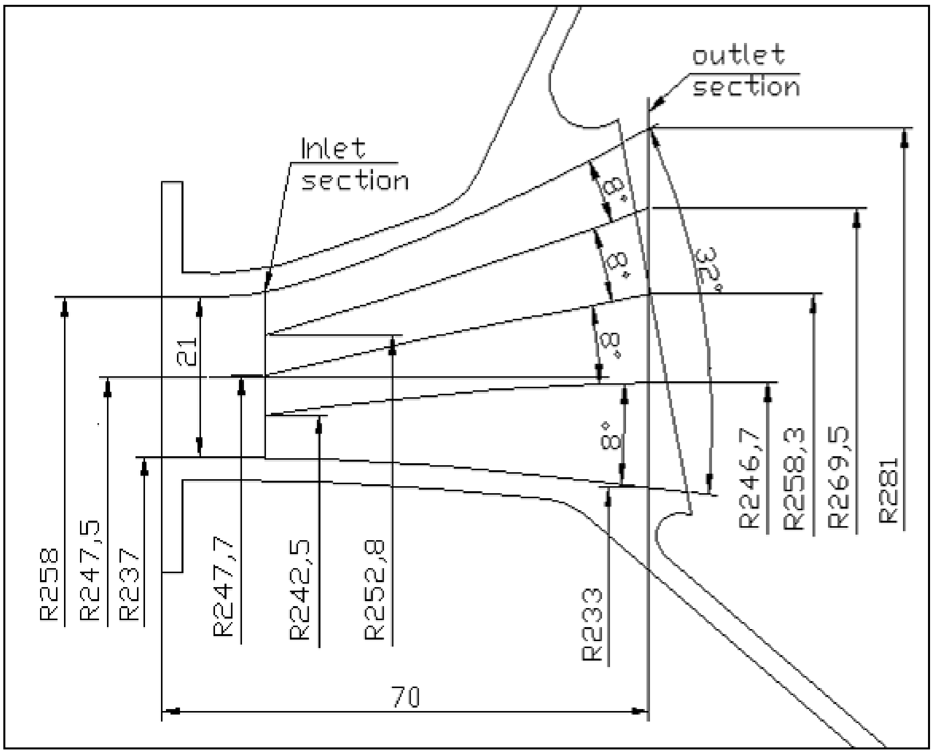


Figure A-1 Four Passages Diffuser Design Results [Khandelwal, 2012 (f)]

Total pressure head has been estimated by the help of Figure A-2 which is about 0.032 when the compressor exit Mach number is 0.23. As a result diffuser total pressure losses are calculated from the following equation [Mohammad, 2009]:

$$\frac{\Delta P}{P_3} = \left[\left(1 - \frac{1}{AR^2}\right) - Cp\right] * \frac{P_3 - p_3}{P_3} = \left[\left(1 - 1/2.3^2\right) - 0.5\right] * 0.032 = 0.00995 \qquad (10)$$

The total pressure loss is calculated as 0.995%.

Casing and Liner

There are two methods commonly used to get the dome and passage sizes using the velocity method and pressure loss method. In this design process the velocity method is selected to calculate the casing and liner dimensions. Initially the air distribution should be calculated. The equivalence ratio of the combustion zone is selected as 0.6, therefore the air flow for combustion with a gaseous fuel is obtained to be 24.68 kg/s which is about 50.86% of the total air mass flow. For cooling considerations the ratio of the dome cooling flow rate to the combustor air flow rate is assumed as 14%. Cooling ratios ranges between 10-15% for a typical combustor [Mohammad, 2009]. Meanwhile the remaining air flow can be used as liner cooling air and dilution air. Mellor [1990] has indicated that the liner convective cooling flow rate is typically 0.75 kg/s m^2 atm. The liner cooling air flow can be assumed as 16% of total air flow before the casing and liner diameter is determined. Once the liner diameter is calculated, it can be checked and adjusted as required. The air distribution is shown in Table A-5.

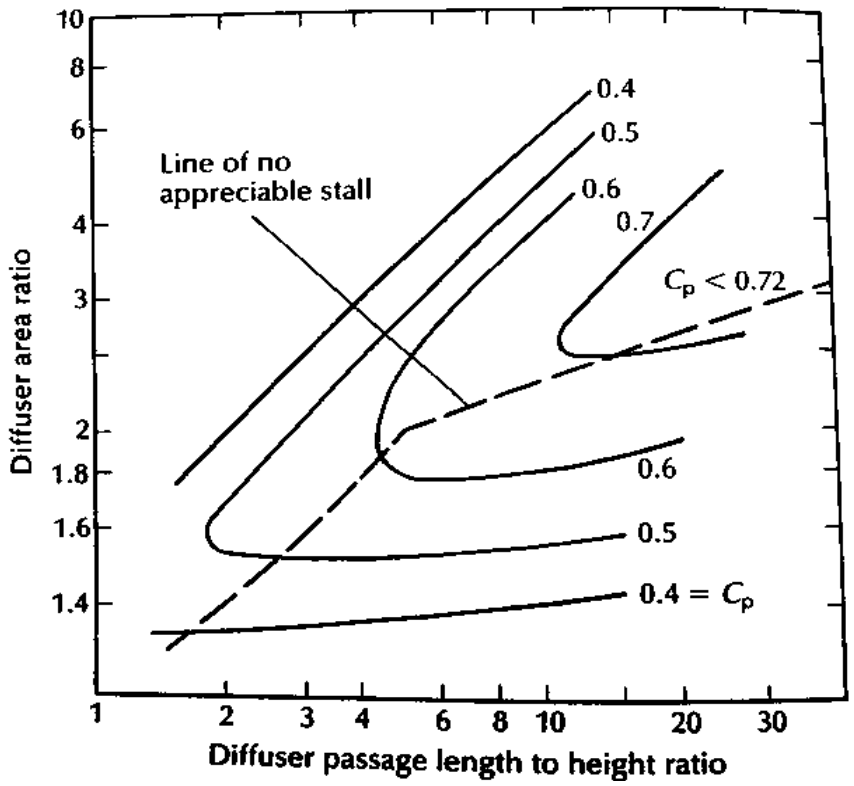


Figure A-2 Two-dimensional Diffuser Pressure Loss [Lefebvre, 2010]

Table A-5 Air Distribution of the Combustor

Zones	Air flow (kg/s)	Percentage (%)
Combustor Total Air Flow	48.53	100
Combustion Air Flow	24.68	50.86
Dome Cooling Air Flow	6.31	14
Film Cooling Air Flow	8.25	16
Dilution Air Flow	9.289	19.14

Secondly the air velocity in the passage and in the dome area should be estimated. Mellor [1990] summarized some typical reference velocities in the dome and passages, which are 35-60 m/s and 7-12 m/s for the passages and dome area respectively. Considering an increase in primary zone volume, the velocity in the dome is assumed to be 8 m/s and the velocity in the passage is assumed as 35 m/s which are lower than the conventional values. Using these velocities the dome area can be calculated from following equation:

$$A_d = \frac{m_d}{\rho_3 * V_d} \tag{11}$$

Where air flow in the dome is the summation of combustion air flow and dome cooling air flow, which is 30.99 kg/s. Similarly, the passage area can be calculated from this following equation:

$$A_p = \frac{m_p}{\rho_3 * V_p} \tag{12}$$

Air flow in the passages is the air flow without the dome air flow, which is 17.54 kg/s.

Using these results the reference area of the combustor (A_{ref}), reference velocity (V_{ref}), dome height (H_d) and overall passage height (H_p) can be calculated from Eqn. 13, 14, 15 and 16 respectively [Mohammad, 2009]:

$$A_{ref} = A_d + A_p \tag{13}$$

$$V_{ref} = \frac{m_c}{\rho_3 * A_{ref}} \tag{14}$$

$$H_d = \frac{A_d}{2 * \pi * r_4} \tag{15}$$

$$H_p = \frac{A_{ref}}{2 * \pi * r_4} - H_d \tag{16}$$

Where r_4 is the outlet section radius of the combustor which is 0.2817m fixed by the engine core of the GDP. The sizes of the casing and liner calculated above are shown in Table A-6. The diameter of the liner can be calculated from the dome height and outlet section radius of the combustor, r_4. Meanwhile the diameter of the casing can be acquired by assuming the areas of the outer passage and inner passage are the same. Referring to the LM6000 combustor, a liner length to dome height ratio of about 1 is acceptable, so the liner length can be obtained by the dome height. The calculated results have been presented in the schematic shown in Figure A-3.

Table A-6 Dimensions of Dome and Passage

	Dome	Passage
Velocity (m/s)	8	35
Air Flow (kg/s)	30.99	17.54
Area (m^2)	0.3329	0.04307
Height (m)	0.188	0.0243

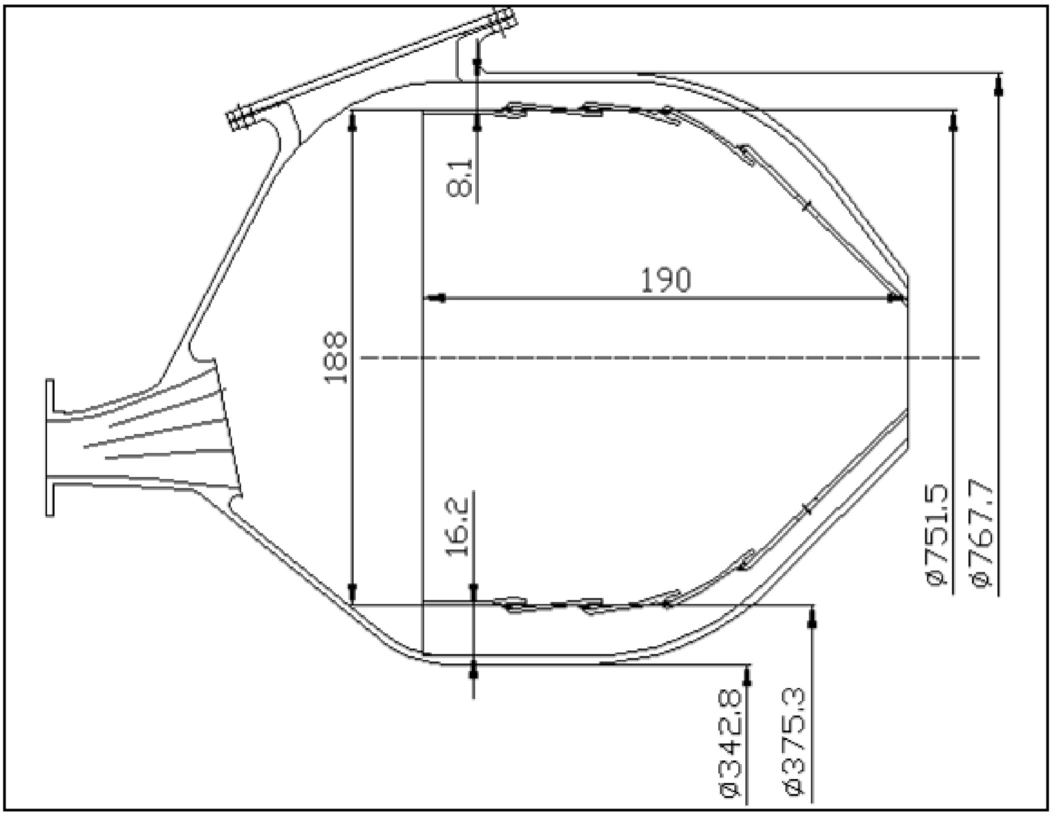


Figure A-3 Dimensions of Casing and Liner [Khandelwal, 2012 (f)]

Fuel Pre-mixer

In this study a Double Annular Counter Rotating Swirler (DACRS) has been selected as the fuel pre-mixer and injector. DACRS has three different variants due to the different fuel injection methods which were introduced by Joshi [1994]. DACRS also works as an axial air swirler because it is a combination of fuel injectors and air swirlers. Initially dimensions of DACRS can be calculated by following the swirler design procedure as DACRS can be considered as a standard swirler. According to Lefebvre [2010], Mellor [1990] and Mohammad [2009], the range of swirler vane angle (θ_v) is 30-60^o. The value of swirler vane angle can be typically assumed as 45^o. It is reasonable to assume that the liner cross sectional area (A_L) is equal to the dome Area (A_d) estimated previously. Also one should notice that the liner pressure drop (pdl) is approximately equal to the swirler pressure drop which is about 60-70% of the total pressure drop in the combustor. k_{sw} is a swirler constant that has a value of 1.3 for flat vanes. m_{sw} is the swirling air flow rate which is equal to the combustion air flow shown in Table A-9. The total swirling air flow is 24.68 kg/s.

The swirler flow path is divided into two passages including an inner flow path and outer flow path. It can be assumed that the outer flow path area of the swirler is around 3 times that of the inner flow path area. This means that the air flowing through the outer flow path is 18.51 kg/s, which is 3 times the flow through the inner flow path. Therefore the area of each swirler flow path (A_{sw}) can be calculated by using Eqn. 17[(8)]:

$$A_{sw} = \frac{1}{\cos(\theta_v)} * \left[\left(\frac{2 * \rho_3 * pdl}{k_{sw} * {m_{sw}}^2} \right) + \frac{1}{{A_L}^2} \right]^{-0.5} \tag{17}$$

It should be noted that the area calculated above is the total area of each flow path. Therefore it should be divided by the number of fuel injectors. Using the staged combustion concept, fuel injection is divided into three stages which form three rings within the annular combustor. Thus each ring has an equal height of one third the height of the dome, which is about 0.062 m. Hence the average fuel injector number of each ring could be estimated as following:

$$N_{fi} = \frac{2 * \pi * r_4}{0.062} \approx 30 \quad (18)$$

The number of fuel injectors in the outer two rings is calculated as 60 in total and the fuel injector number of inner ring is 15 due to its smaller radius. Hence the total number of fuel injectors is 75, which is as the same as that of LM6000 engine combustor. Assuming the flow coefficient of the swirler is 0.7, the actual flow area of each swirler flow path (A_{sw}') can be calculated by Eqn. 19:

$$A_{sw}' = A_{sw} / (0.7 * 75) \quad (19)$$

Next the hub diameter of the inner flow path (D_{hub}) is selected as 0.01 m. Requiring the tip diameter (D_{tip}) of the swirler inner flow path to be calculated from this equation(8):

$$A_{sw}' = \frac{\pi}{4} * \left(D_{tip}^{\ 2} - D_{hub}^{\ 2}\right) - 0.5 * n_v * v_t * \left(D_{tip} - D_{hub}\right) \quad (20)$$

Where v_t is the vane thickness which ranges 0.7-1.5 mm, v_n are the number of vanes whose range is from 8-16.

Hence the swirl number for the swirler (SN) which should be larger than 0.6 can be checked from following equation [Lefebvre, 2010]:

$$SN = \frac{2}{3}\left(1 - \left(\frac{D_{hub}}{D_{tip}}\right)^3\right) / \left(1 - \left(\frac{D_{hub}}{D_{tip}}\right)^2\right) * \tan(\theta_v)$$

(21)

The vane spacing (vsp) is estimated using Eqn. 22 [Mohammad, 2009]. The vanes chord (v_{ch}) is estimated as one to two times of vane spacing.

$$\mathrm{v_s} = \frac{\pi * \left(D_{tip} + D_{hub}\right)}{2 * n_v} \quad (22)$$

The design results are obtained and illustrated in Table A-7.

The next step is to determine the fuel injection method and area of the injection holes. According to Joshi et al. [1994], the fuel injection holes could be located at the center of the pre-mixer body through which the fuel flows. Otherwise be located at the fuel injection spokes which is just behind the trailing edge of the vanes and is supplied by the centre body fuel flow. The third method of fuel injection is fuel flow within the outer vanes that is injected by small holes at the trailing edge of vanes and the fuel is supplied by the circumferential fuel manifold.

Table A-7 Design Results of Swirler

NAME	Inner Flow Path	Outer Flow Path
A_L ,Liner cross sectional area (m^2)	0.3329	
pdl, Liner pressure drop (Pa)	73318.2	
θ_v, Vane angle (^{o})	45	45
k_{sw}, Swirler constant	1.3	1.3
m_{sw}, Air flow rate (kg/s)	6.17	18.51
A_{sw}', Actual area (m^2)	0.01464	0.04389
n_v, Vane thickness (m)	0.001	0.003
v_n, Number of vanes	8	8
Dhub, Hub diameter (m)	0.01	0.023
Dtip, Tip diameter (m)	0.02	0.039
SN, Swirler Number	1.26	1.31
v_{sp}, Vane spacing (m)	0.0058	0.012
v_{ch}, Vanes chord (m)	0.012	0.012
Width of vanes	0.64	0.64

In this study the third fuel injection method is selected because it can supply a better mixture of fuel and air. It should be noticed that there are three arrangements about the fuel injection holes, introduced by Joshi et al. [Joshi, 1994].

Figure A-4 shows the three different configurations of fuel injection holes. Ekstedt [1994] summarized that the diameter of the fuel injection holes are approximately 0.6 millimeters in order to minimize plugging therein maximizing air fuel mixing. The number and size of fuel passages in vanes are dependent on the amount of fuel flow through the fuel manifold, the fuel pressure, and the particular design of swirler vanes. However, it has been found that three fuel passages can work successfully and adequately. Therefore to simplify the design of the pre-mixer the fuel injection through the outer vanes is selected as the injection method. The diameter of the fuel injection holes are 0.6 mm and each outer vane has three fuel passages, as suggested by Ekstedt [1994].

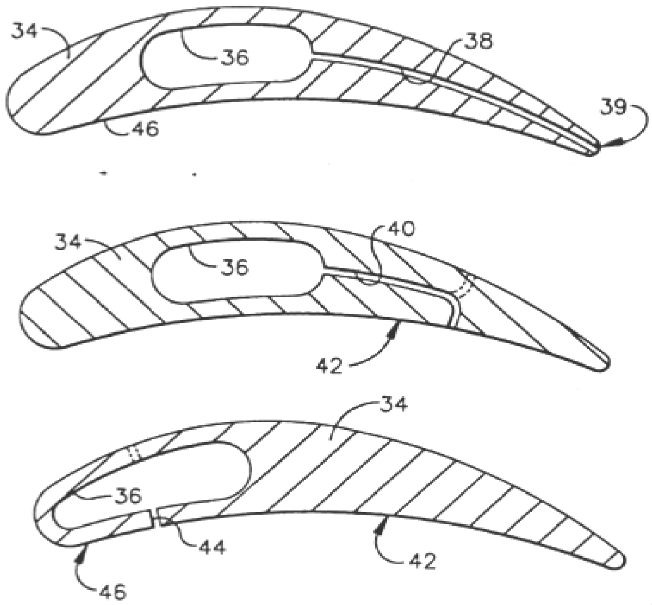


Figure A-4 Fuel Injection Methods through Swirler Vanes [Joshi, 1992]

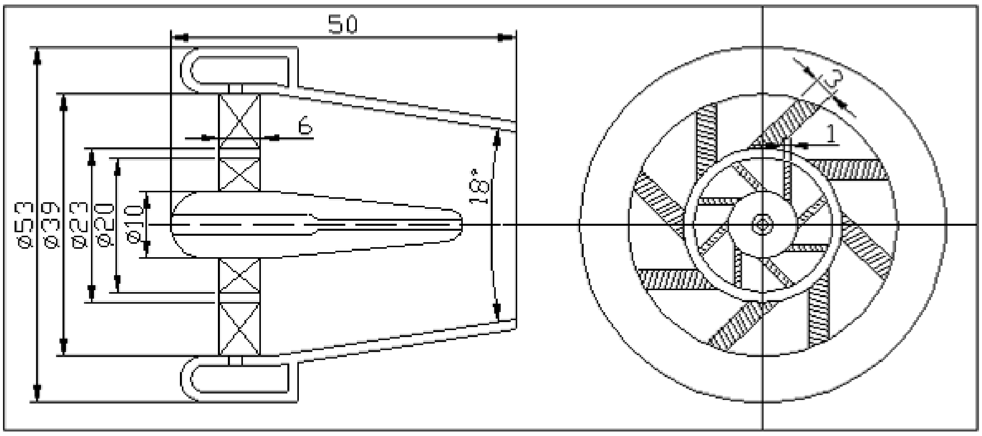


Figure A-5 Design Results of DACRS [Khandelwal, 2012 (f)]

The designed results of DACRS are shown in Figure A-5. The fuel injectors are divided into three stages. Hence each ring of the outer two rings has 30 fuel injectors while the inner ring has 15. The arrangement of fuel injectors are shown in Figure A-6.

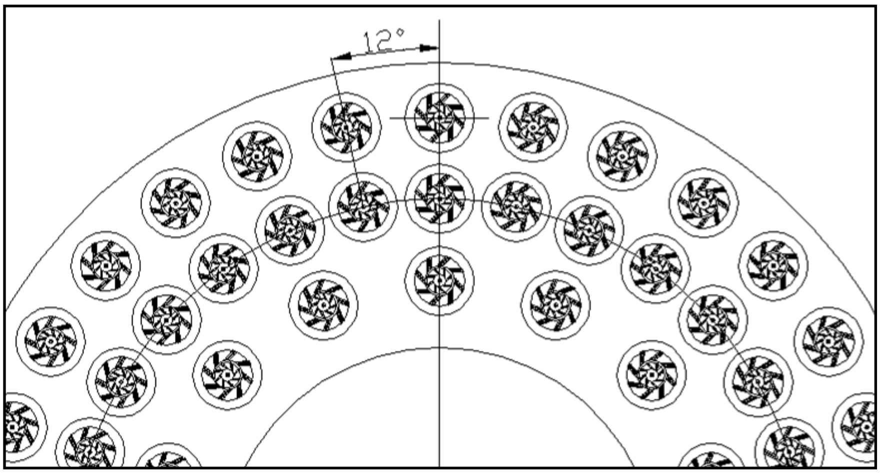


Figure A-6 Arrangement of DACRS [Khandelwal, 2012 (f)]

COOLING SYSTEM

The cooling system consists of two techniques. One is dome cooling and the other is the liner cooling. In this section both cooling methods will be designed.

Dome Cooling

The heat shield is located at the end of the fuel pre-mixer at the sides of the combustion region. This can form a cool area between the combustion zones. To improve its cooling effectiveness some ribs are made use of inside the cooling air flow and the air is jettisoned at the end of the heat shield through a ring of small holes [Ekstedt, 1994]. The LM6000 combustor utilizes four heat shields because it does not adopt the liner cooling system in its design. The dome cooling air flows within the heat shield and perpetrates into the combustor chamber through multiple rings of holes. Figure A-7 shows the location of the heat shield. In this gas turbine combustor design the liner cooling system is employed in order to effectively cool the liner so that only two heat shields are required. Hence a portion of dome cooling air will be employed for heat shield cooling and the remaining cooling air will be supplied to the outer and inner areas of the dome plates. Figure A-7 shows the layout of the dome cooling method. Generally the area of the coolant metering holes can be calculated for meeting the mass flow rate requirements, using a simple incompressible orifice equation [Mellor, 1990]:

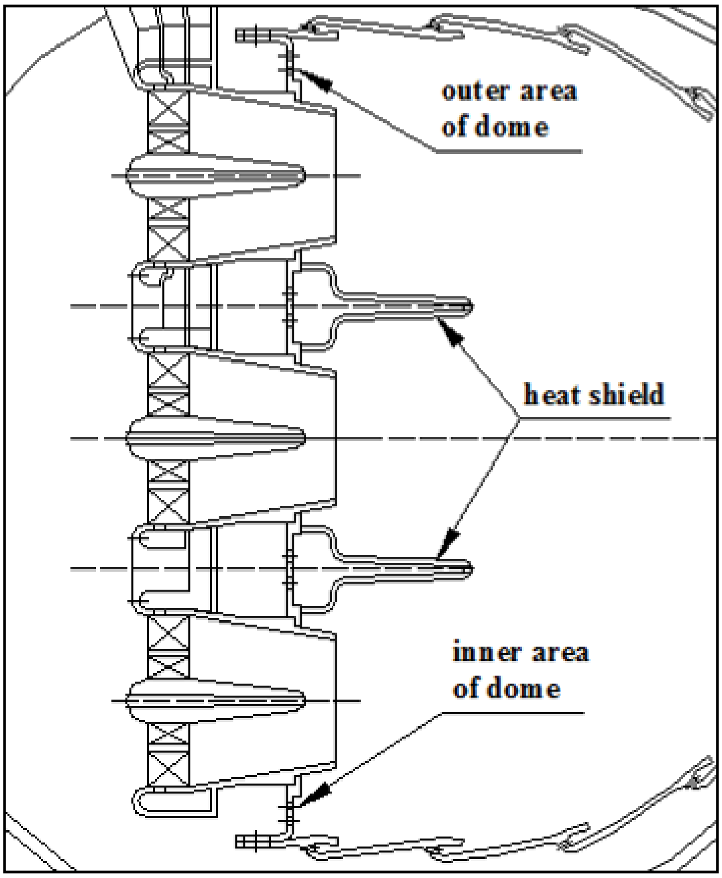


Figure A-7 Dome Cooling [Khandelwal, 2012 (f)]

$$A_{\text{hole}} = \frac{W_{hole}}{Cd * \sqrt{2\rho \Delta P}} \tag{23}$$

Pressure drop at the dome region is usually around 60-70% of the total pressure drop in the combustor chamber. The discharge coefficient (C_d) of holes depends on diameter, metal thickness and method of drilling of the holes. Commonly, C_d = 0.8 and is used for initial estimates. The calculated results are listed in Table 8. After the flow area is obtained, the diameter and number of cooling holes can be calculated from the following equation [Mellor, 1990]:

$$D_{\text{hole}} = \sqrt{\frac{4 * A_{hole}}{\pi * N_{hole}}} \tag{24}$$

As shown in Figure A-7, one ring of cooling holes is located at each heat shield. Two rings of cooling holes will be implemented at the outer section of dome plates and two rings are also at the inner section. The spacing between the two holes is selected as 0.004 m and the average diameter of each cooling hole ring can be calculated based on the dimensions of the liner and pre-mixers. The number of holes in each ring and the total number in the dome regions can be calculated. The diameter of cooling holes can be obtained from Eqn. 24. Finally the calculated results of the dome cooling holes are shown in Table A-9. Figure A-8 to A-10 shows the design results and the arrangement of the dome cooling holes.

Table A-8 Design Result of Cooling Holes Area

W_d, Dome Cooling Air (kg/s)	C_d, Hole Discharge Coefficient	ρ, Air Density (kg/m^3)	ΔP, Dome Pressure Drop (Pa)	A_{hole}, Cooling Holes Area (m^2)
6.31	0.8	11.63	85537.9	0.00603

Table A-9 Dome Cooling Holes Results

	Inner Heat shield	Outer Heat shield	Inner dome plate	Outer dome plate
Rings of Holes	1	1	2	2
D, Average Diameter (m)	0.504	0.624	0.39	0.74
B_{hole}, Spacing (m)	0.004	0.004	0.004	0.004
n_{hole}, Number of Holes	396	490	612	1160
D_{hole}, Diameter of Holes	0.00163			

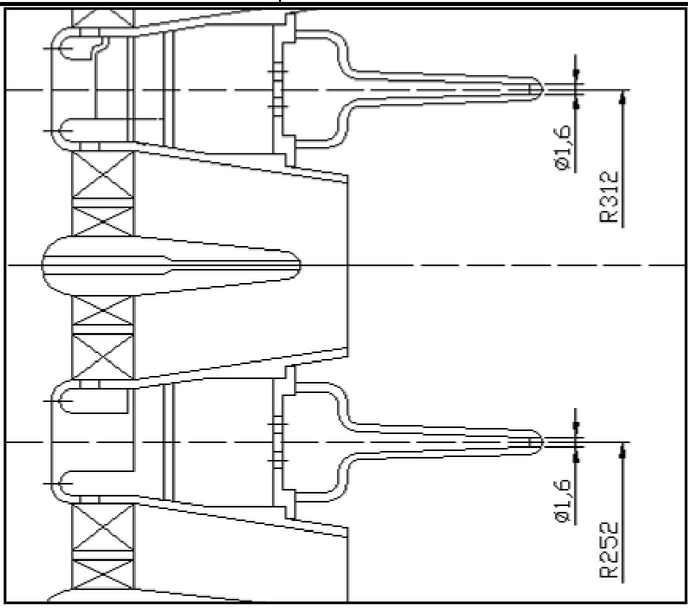


Figure A-8 Dimensions of Cooling Holes at Heat Shield [Khandelwal, 2012 (f)]

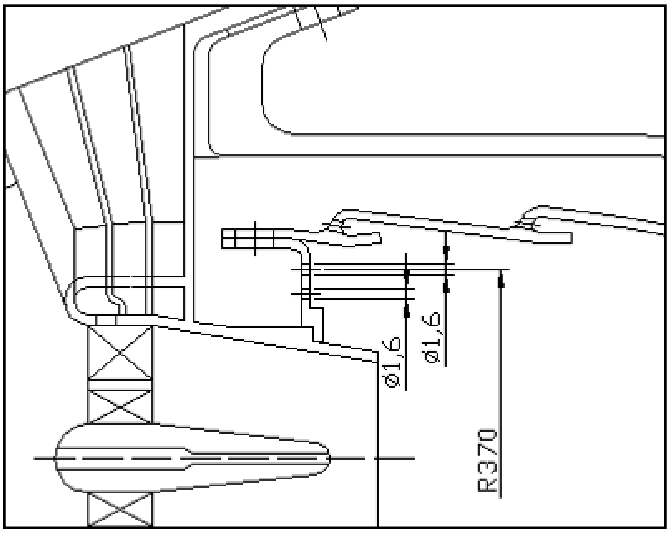


Figure A-9 Dimensions of Cooling Holes at Outer Dome Plate [Khandelwal, 2012 (f)]

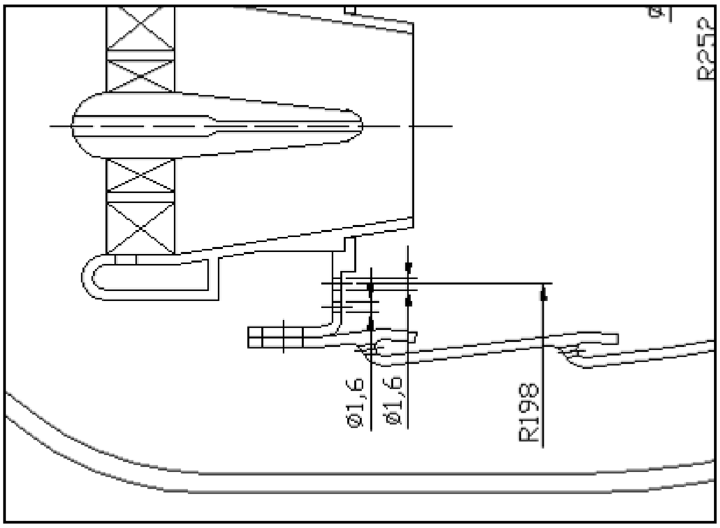


Figure A-10 Dimensions of Cooling Holes at Inner Dome Plate [Khandelwal, 2012 (f)]

Film Cooling

The other cooling system used is the film cooling in the liner. In this study the concept of a double cooling wall is employed. Figure 11 shows a typical double cooling wall system widely used in modern gas turbine combustors. D is the diameter of cooling holes, B is the spacing between every two holes, L is the mixing chamber length, H_{in} is the height of the mixing chamber at the coolant inlet section and H_{ou} is the height of the mixing chamber at the coolant outlet sections. The typical values of these parameters are list in Table 10. The next step is to calculate the physical area of the cooling holes (A_{hole}) from Eqn. 23. The liner pressure drop is typically 3-5% of inlet pressure. The results are listed in Table 11.

Table A-10 Typical Values of Slot Geometry [Mellor, 1990]

Parameter	Definition	Typical Values (m)
D	Hole diameter	0.001-0.0025
B	Pitch between metering holes	0.0025-0.0075
L	Internal flow path length	0.002-0.006
I	Length of coolant jet prior to impingement	0.0015-0.004
H_{in}	Internal height of slot mixing chamber	0.0015-0.004
H_{ou}	Slot height	0.0015-0.0025

Table A-11 Design Result of Cooling Holes Area

W_f, Film Cooling Air (kg/s)	C_d, Hole Discharge Coefficient	ρ, Air Density (kg/m^3)	ΔP, Liner Pressure Drop (Pa)	A_{hole}, Film Cooling Holes Area (m^2)
8.25	0.8	11.63	82380	0.007448

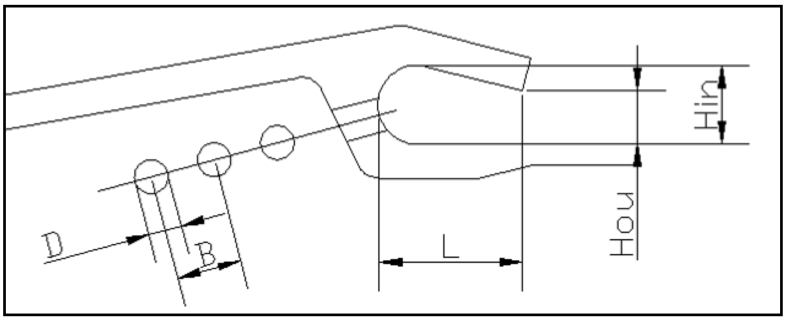


Figure A-11 Typical Double Cooling Wall [Mellor, 1990]

Referring to the LM6000 combustor whose liner length to dome height ratio is about 1 with increased volume, the liner length to dome height ratio of this combustor is selected as 1 also. Therefore the liner length is about 0.18 m. The typical distance between the two rings of film cooling holes are in the range of 0.03m to 0.06m so the distance is assumed as 0.03m. Hence four stages of film cooling holes can be arranged in each liner. The liner consists of an inner liner and an outer liner. Permitting four rings of holes that are at the inner liner with an average liner diameter of 0.375m while four rings are at the outer liner with an average liner diameter of 0.750m. The final design results of film cooling holes are obtained from Eqn. 24 and listed in Table A-12. To design an effective film slot, the MIX_N parameter must be observed, presenting the relative merit for the slot geometry and coolant metering hole pattern. The MIX_N parameter can be calculated by [Mellor, 1990]:

$$MIX_N = \frac{B * H_{ou} * I}{D_{hole} * H_{in} * L} \qquad (25)$$

Table A-12 Film Cooling Holes Results

Name	Inner liner	Outer liner
Rings of Holes	4	4
D, Average Diameter (m)	0.375	0.750
n_{hole}, Holes Number of Each Ring	336	675
B_{hole}, Spacing (m)	0.0035	
D_{hole}, Diameter of Holes	0.00153	

Therefore the MIX_N parameter is obtained when the geometry variables are selected which are shown in Table A-13. A good practical value of the MIX_N parameter is less than 0.5, so the design of this slot system is acceptable as the MIX_N parameter is 0.457. The design results of the double wall cooling system are shown in Figure A-12 and Figure A-13.

Table A-13 Geometrical Variables of Film Cooling

Name	Value
L, Internal flow path length (m)	0.006
I, Length of coolant jet prior to impingement (m)	0.002
H_{in}, Internal height of slot mixing chamber (m)	0.0025
H_{ou}, Slot height (m)	0.0015
MIX_N parameter	0.457

Dilution Holes

About 80% of the total combustor air flow is used in combustion in the LM6000 combustor and there is no film cooling configuration in this combustor due to the application of thermal barrier coatings. Therefore, nearly 20% of the total combustor air flow is used as dilution air. In this design task about 19% of the total combustor air flow is used to dilute the combustion products. There are two design methods widely used to design the dilution holes. One is the Cranfield design method and the other is the NASA design method. In this design process, the Cranfild design method is used to size the dilution holes. Lefebvre(1) has summarized the Cranfield design method and supplied an equation to calculate the effective diameter, D_e, as follows:

$$\mathrm{n}D_e^{\,2} = 15.25 * m_{dilu} * \left(\frac{P_3 * \Delta P_L}{T_3} \right)^{-0.5} \tag{26}$$

n is the number of dilution holes.

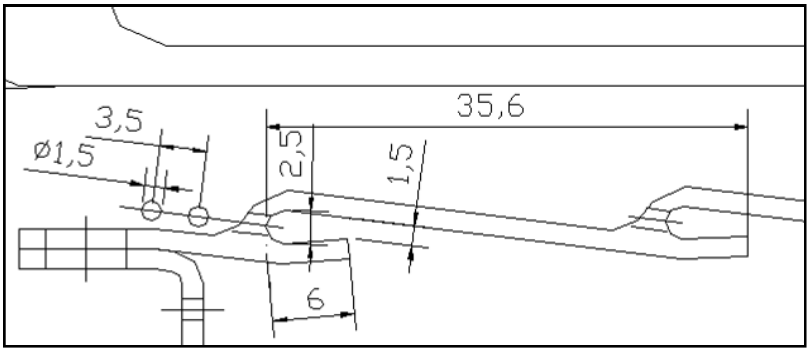


Figure A-12 Design Results Double cooling wall [Khandelwal, 2012 (f)]

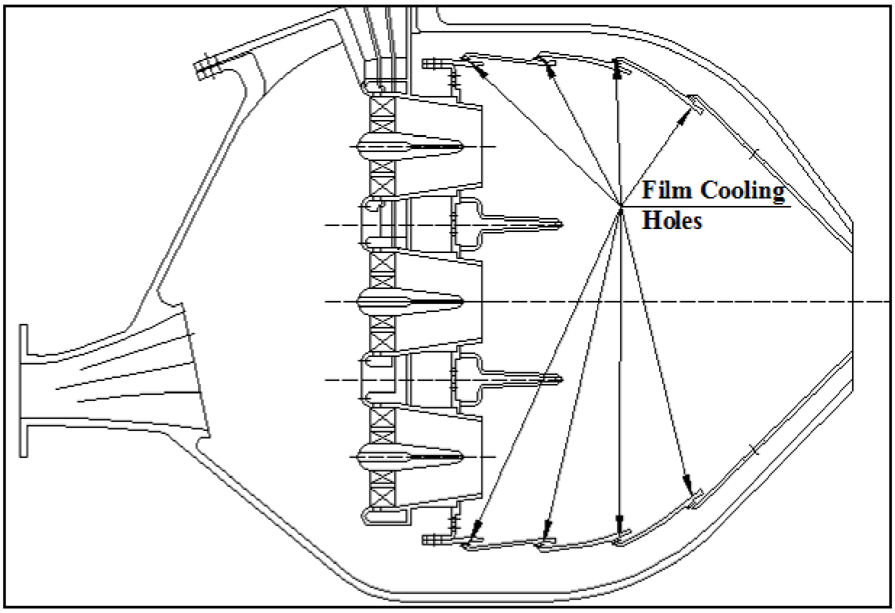


Figure A-13 Arrangement of Film Cooling Holes [Khandelwal, 2012 (f)]

According to Mohammad [2009], it can be assumed that each ring of dilution holes have two holes per fuel injector. This means that there are 120 dilution holes in total on the inner and the outer liner. Dilution mass flow, total pressure, liner pressure drop and total temperature have already been calculated. Hence the effective diameter of the dilution holes, De, has been calculated as 8.3 mm and the actual diameter is calculated as 12 mm. Next the actual geometrical diameter of the dilution holes (D_{hole}) can be obtained by:

$$D_{hole} = \frac{D_e}{C_d^{\ 0.5}} \qquad (27)$$

Where C_d is the hole discharge coefficient and for the preliminary design an average value of 0.5 is acceptable [Lefebvre, 2010]. The design results of dry low emission industrial combustor are shown in Figure A-14.

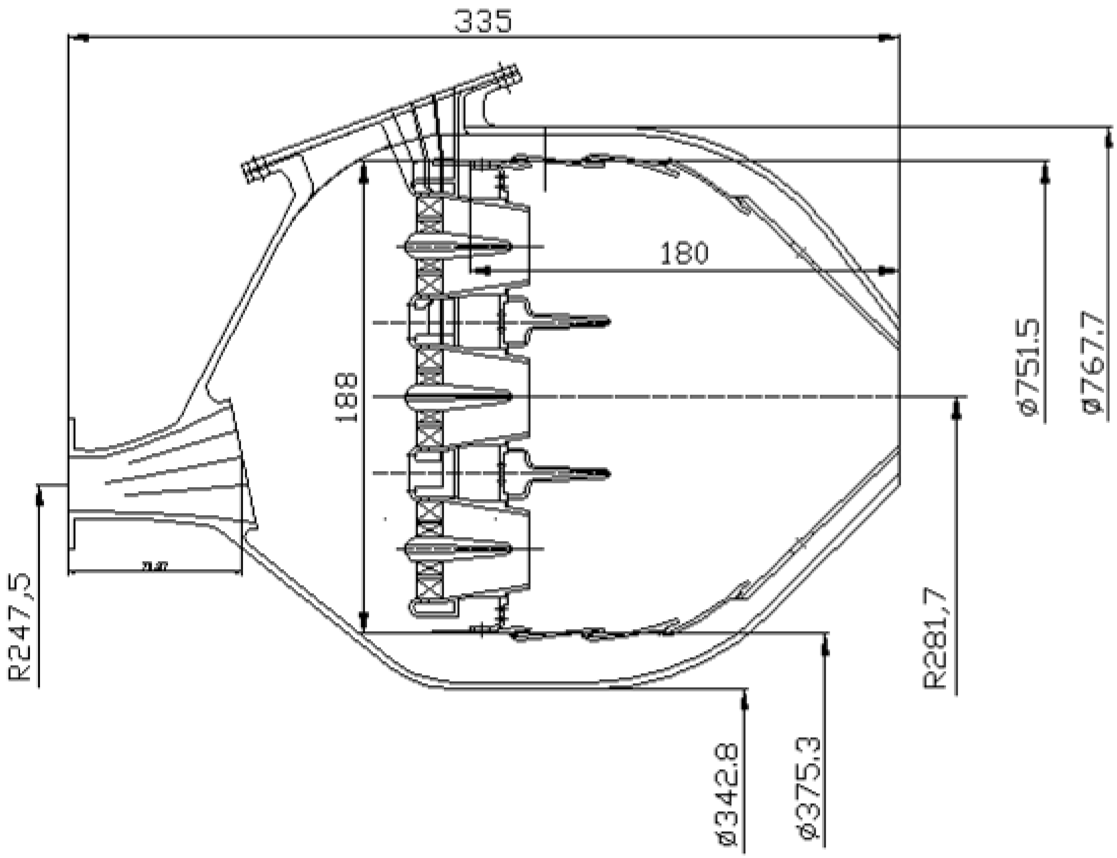


Figure A-14 Design Results of Industrial Combustor [Khandelwal, 2012 (f)]

A.1.4 Performance Evaluation

Fuel Injection Quality

According to Lefebvre [2010], gaseous fuels present no specific problems from a combustion standpoint having high energy densities. They are usually characterized by clean combustion with low level of formations of soot and oxides of nitrogen. For liquid fuels the mean drop size of fuel spray droplets are an important criterion and many equations have been developed to predict the fuel spray quality when a fuel injector is designed. However gaseous fuels do not have any such spray problems as there is no need to evaporate the fuel because it is already in the form of a gas. Joshi et al. [1994] has summarized that the fuel jet integration is unsatisfactory when fuel is injected from the centre body of DACRS. Hence a modification was made by adding radial spokes in the location of these holes in the centre body. The combustion test of this configuration showed that single digit NO_x emissions are attainable similar test results can be obtained when fuel is injected from the hollow outer vanes. These DACRS are applied to a range of GE engines including LM1600, LM2500 and LM6000. Single digit NO_x emissions have been attained at test conditions surrounding the operating ranges of these engines.

Using the third configuration of DACRS which provides fuel from hollow outer vanes, the gaseous fuel is injected into swirled air at the trailing edge of the outer vanes of the swirlers and immediately mixed with air so that a lean homogeneous fuel to air mixture can be formed before combustion. The quality of the fuel to air mixture of this design can be regarded as proficient because the fuel injector is designed from the concept of DACRS. The arrangement of the fuel pre-mixers can provide a good quality of fuel to air mixture in the whole combustion zone, which is similar with LM6000 combustor developed by GE.

NO_x Emission

Many empirical and semi-empirical models are widely used for correlating the experimental data on pollution in terms of all the relevant parameters of combustion. These include combustor dimensions, design features, operating conditions, fuel type and fuel spray quality. Hence, the empirical models can play an important role in the design and development of a low emission combustor [Lefebvre, 1981]. The emissions are mainly dependent on three terms including mean residence times in the combustion zone, chemical reaction rates and mixing rate. Several researchers have proposed different correlations for predicting NO_x emissions. Odgers and Kretschmer [1985] provided a correlation for predicting NO_x emissions as follows:

$$\mathrm{NOx} = 29 \times \exp\left(-\frac{21670}{T_C}\right) \times P_3^{0.66} \times \left[1 - \exp\left(-250\tau\right)\right] g/kgfuel \qquad (28)$$

When using Eqn. 28 it should be noted that the time for NO_x formation for aircraft combustors is 0.8 ms (airblast atomizers) and 1.0 ms (pressure atomizers). While time for NO_x formation in industrial combustors burning liquid fuels range from 1.5 to 2 ms. This equation may be used for an industrial combustor where the NO_x formation time is assumed.

Lewis [Lewis, 1991] supplied another correlation:

$$\mathrm{NOx} = 3.32 \times 10^{-6} \exp\left(0.008 T_C\right) P_3^{0.5}\, ppmv \qquad (29)$$

Eqn. 29 is satisfactory with the correlative measurements of NO_x emissions when lean homogeneous mixtures are supplied with gaseous fuels. Therefore Eqn. 28-29 is selected to predict the NO_x emissions of the industrial combustor designed in this study. It should be taken into consideration that for most industrial combustors using gaseous fuels, the combustion temperatures are lower than conventional liquid fuel combustors when they operate at lean fuel conditions. The combustion temperature is assumed as 1900°K [Lefebvre, 2010].

Frazier et al. [2001] has reported the DACRS can reduce NO_x emission significantly. The NO_x emissions can be decreased approximately by 60% when the equivalence ratio is decreased from 0.7 to 0.6 with DACRS. NO_x emissions are measured as low as 3.8 ppmv when the inlet pressure of the combustor is 114.2 kPa. The pressure drop of the combustor is about 5.1%, the equivalence ratio is 0.6 and the inlet temperature is 592°K. Badeer [Badeer] has reported that the triple annular combustor concepts can largely cut down NOx emissions also. It is found that when gaseous fuels are used the NO_x emission can be reduced largely when DLE technologies are applied to modify single annular combustors. The NO_x emissions typically range from 127 to 205 ppmv when the combustors are in the single annular configuration and are reduced to 25 ppmv when DLE technology is utilized.

The operating conditions of this combustor are similar with Frazier et al. [2001] test conditions; having an equivalence ratio of 0.6 with an inlet temperature and pressure slightly higher than tested. Finally it is reasonable to predict that the NOx emissions will range between 21.52 ppmv and 69.02 ppmv, which have been calculated by Eqn. 28 and 29 respectively. These results are based on empirical and semi-empirical correlations which are not accurate enough. Although the results do give a good representation of NO_x emission levels for this industrial combustor design, showing the incentive of reduced NO_x emissions.

CO Emissions

Similar correlations for CO emissions are observed using calculation that have been developed by Lefebvre [1984], and Rizk and Mongia [1994]. In their correlations the relevant temperature is not the local peak value adjacent to the evaporating fuel drops; but it is the average value throughout the primary zone (T_{pz}) which is lower than the peak local temperature due to the application of cooling systems.

$$CO = 86\mathrm{m}_3 T_{pz} \frac{\exp\left(-0.00345 T_{pz}\right)}{V_c - V_e}\left(\frac{\Delta P_{3-4}}{P_3}\right) P_3^{1.5} g / kgfuel \tag{30}$$

$$V_e = 0.55 m_{pz} D_0^{\ 2} / \rho_{pz} \lambda_{eff} \tag{31}$$

Where V_e is proportional to the square of the initial mean drop size.

This equation highlights the importance of good atomization to the attainment of low CO emissions. However this correlation is also suitable for liquid fuel combustors. These equations are more likely to be used to predict the CO emissions of liquid fuels. On the contrary it is reasonable to believe that the CO emissions from this combustor will be lower than that from common industrial combustors. This is because the volume of the primary zone is almost twice that of common combustors. Form tests and operational results from the LM6000 combustor [Frazier, 2001], the CO emissions are checked to be cut down with an increase of the primary zone volume as the residence time of fuel to air mixture are increased.

A.1.5 Conclusion

A triple annular dry low emission industrial combustor is designed which is similar to the LM6000 combustor. Advanced DLE methods such as lean fuel combustion concepts with the pre-mixed method, staged combustion concepts, triple annular arrangements, multi-passage diffuser designs, double wall cooling systems, double annular counter rotating swirlers (DACRS) and heat shields are employed to reach the intent of cutting down the emissions.

The design process is shown step by step for the combustor design, with a number of experienced values and methods used in this system. To begin with the diffuser is designed as a multi-passage diffuser which consists of four passages divided by three splitter vanes. The pressure loss of this diffuser is about 0.99%. The casing and liner are sized by the velocity method based on the operating conditions of the Cranfield's 2010 GDP combustor. When the equivalence ratio is chosen as 0.6 the volume of combustor is nearly two times that of the GDP aero combustor. The flow area of the fuel injectors are calculated when the air flow that passes through it is determined. Because the fuel injector is a DACRS it works as a combination of fuel injector and swirler. A conventional swirler model is employed as the design method for determining the dimensions. The diameter of the fuel injection holes adopts an established value of 0.6 mm and each outer vane has three injection holes. For a staged combustion purposes the DACRS are arranged in three rings in front of the primary zone having a total of 75, consisting of two rings of 30 and an inner ring of 15 DACRS. The cooling system is designed with both in dome cooling and liner cooling. Heat shielding is adopted between fuel injectors cooling the dome area, which has four rings of cooling holes that are arranged on the dome plates. The double wall cooling method is employed to reduce the temperature of the liners. Both the diameters of dome cooling and liner cooling holes are calculated based on correlation among flow area, cooling air flow rate, air density and pressure drop. Finally the dilution holes are determined by the Cranfield method.

As a result the performance of this DLE combustor is predicted by empirical correlations. Three calculation methods are used to predict the NOx emissions of this combustor. The NOx emissions are about 20 to 70 ppmv which is about 10% to 30% of conventional combustors. For an industrial combustor burning gaseous fuels the NOx emissions could reach a value as low as 20 ppmv or even lower. This will result in NOx emissions being decreased up to 90% compared with conventional single annular combustors. The calculated results are acceptable. Hence, the design purpose of low pollutant emissions is achieved due to the application of advanced dry low emission methods.

A.2 Double Annular Combustor

A.2.1 ABSTRACT

The most uncertain and challenging part in the design of a gas turbine has long been the combustion chamber. There has been large number of experimentations in industries and universities alike to better understand the dynamic and complex processes that occur inside a combustion chamber. This study concentrates on gas turbine combustors as a whole, and formulates a theoretical design procedure for staged combustors in particular. Not much of literatures available currently in public domain provide intensive study on designing staged combustors. The work covers an extensive study of design methods applied in conventional combustor designs, which includes the reverse flow combustor and the axial flow annular combustors. The knowledge acquired from this study is then applied to develop a theoretical design methodology for double staged (radial and axial) low emission annular combustors. Additionally a model combustor is designed for each type; radial and axial staging using the developed methodology. A prediction of the performance for the model combustors is executed. The main conclusion is that the dimensions of model combustors obtained from the developed design methodology are within the feasibility limits. The comparison between the radially staged and the axially staged combustor has yielded the predicted results such as lower NO_x prediction for the latter and shorter combustor length for the former. The NO_x emission result of the new combustor models are found to be in the range of 50-60ppm. However the predicted NOx results are only very crude and need further detailed study.

A.2.2 INTRODUCTION

The combustor design for gas turbines is long been considered as a black art or art more than science. This impression persists mainly because of the empirical nature of design method followed widely. Over the years, large amount of experimental data gathered from research carried out in industry and universities across the world has enabled one to develop a theoretical design methodology for design of a gas turbine combustor. The general procedure for a classic combustor design is show in a step by step process in Fig. A-15.

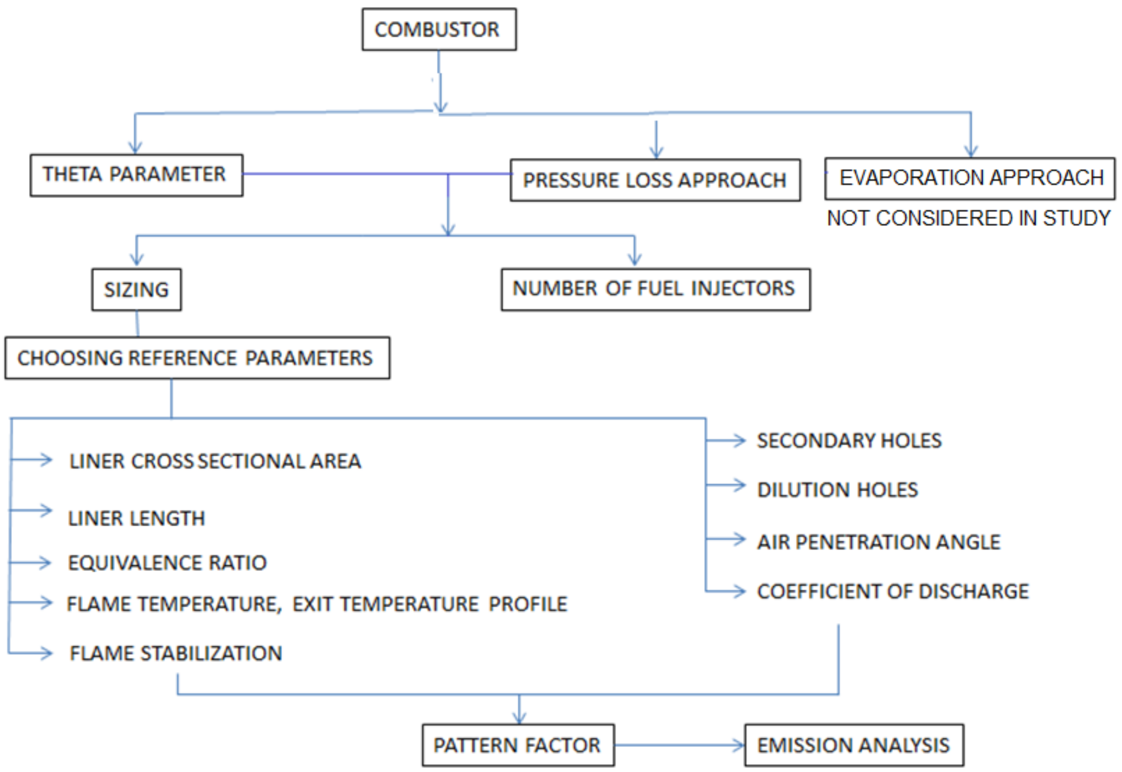


Figure A-15 Flow chart of classic combustor design methodology

In general, gas turbine combustors are sized based on their application. That is, to design a combustor for flying applications, the most important criteria is reliable altitude relight characteristics whereas to design a ground based gas turbine combustor, minimum pressure loss is most desired. For preliminary design of dome and getting reference flow velocities theta parameter approach is used for flying applications and pressure loss approach is used for ground based applications [Mohammad, 2009].

In case of jet engines, combustor sizing is done using both methods and the larger of the two is chosen. This process is followed to maintain maximum reliability during altitude relight. Recently a large number of combustor design methodologies have been reported in literature [Mohammad, 2009; Khandelwal, 2011 (a); Mellor, 1990 (a)].

The greater usage of fossil fuel in the recent years and its associated ill effects has pushed newer combustor designs to be more and more environmentally friendly. The limitations of wet and dry methods of achieving low emissions from combustors have led to the development of staged combustors, in which, no attempt is made to achieve all the performance objectives in a single combustion zone. Instead two or more zones are employed, each of which is designed particularly to optimise certain aspects of combustion performance. There are two ways of staging the fuel injection in combustion chambers; either in radial direction (parallel staging), or in axial direction (series staging) each of which is designed particularly to optimise certain aspects of combustion performance. Staged combustion is one of the most actively researched techniques proposed to reduce pollutants for aero and industrial gas turbine engines. Staged combustion is also termed as dry low emissions (DLE) since there is no need for the injection of water or steam to achieve a significant reduction in pollutant levels (wet method). This makes staged combustion more economical and viable for an aero gas turbine engine. Since staged combustors are comparatively simple and just an extension to the conventional ones, there has been considerable interest and effort invested on them. There are many industrial burners from GE and Rolls Royce with a successful implementation of staged burning concept. However, to the author's knowledge, no literature has been reported to explain the preliminary design procedure of staged combustors. In this study an extensive preliminary design methodology have been presented for staged combustors.

A.2.3 DESIGN METHODOLOGY

The major difference between the design procedure of single annular combustor (SAC) and double annular combustor (DAC) lies in the sizing of the dome. For SAC, the dome area is directly proportional to the primary zone equivalence ratio. The dome size increases with the reducing equivalence ratio and vice-versa. This means to say, that as the value of ϕ reduces e.g. 09 or 08 or so on the dome area increases to accommodate the extra air needed for lower ϕ values and as it goes up e.g. 1 or 1.1 or so on, the dome area reduces and higher quantity of air is admitted through intermediate or tertiary zones. The combustor length is deduced from simple arithmetic, from the combustor volume calculated on the basis of combustion efficiency and the dome height calculated from the method described above.

In designing a DAC, the value of ϕ is very low (overall for the primary region). This yields a very large dome size. From the calculated dome size, two different domes are designed. These are one for the pilot and the other for the mains fuel supply. This breaks down again in to calculating the ϕ value for each of them and then sizing the dome accordingly. Now using the combustor volume which is pre calculated from the efficiency and relight conditions, and also the dome area, combustor length is set. Although there may be different ways of achieving stages combustor design, the intent of the paper is not go in depth about the differences in the design methodologies as the authors attempt only to produce preliminary sizing for combustors. This work concentrates on designing DAC by stretching the classical design methods, and comparing the final model with that of the existing DACs to validate the theory. There by, simplifying the method to achieve the same result.

The design methodology developed in this study follows the conventional method to certain extent and then the idea is extended to accommodate the second stage while keeping in mind the principal changes that follow in terms of change in the combustor cross sectional area, the overall equivalence ratio for low emissions and also the placement of swirlers and the number of fuel injectors. The design splits the combustor into parts and each part is individually designed and assembled to form the final design. The assumptions made in designing the combustor based on the methodology developed are on the basis of one existing dual annular combustor. The model combustor is designed on the basis of available data, simplicity and easiness in adaptability in to any code.

The design methodology proposed here is a collection of previously used and published methods, however, to the best of authors' knowledge a paper consisting of all the steps used as in this literature has not yet been released in to public domain. The design methodology follows a two-fold approach. The first step is overall combustor sizing. The next step is detailed design of components and zones. The basic combustor sizing follows the classic combustor design methodology and this forms the overall dimensions of the combustor.

A.2.4 OVERALL COMBUSTOR SIZING

The compressor exit flow conditions and the geometrical conditions are taken as the input for calculation, γ, ϕ, dome and passage velocities are assumed. The basic calculations are done on the basis of following steps.

The static temperature t_3, static pressure p_3, density ρ_3, velocity V_3 at the diffuser inlet can be calculated using the following aerodynamic equations:

$$t_3 = \frac{T_3}{1+\frac{\gamma-1}{2}*M_3{}^2} \tag{32}$$

$$p_3 = (t_3/T_3)^{\gamma/(\gamma-1)} * P_3 \tag{33}$$

$$\rho_3 = \frac{p_3}{R*t_3} \tag{34}$$

$$V_3 = M_3 * \sqrt{\gamma * R * t_3} \tag{35}$$

The inlet area of the diffuser A_3 and the inlet width of the diffuser H_{diff} can be calculated as follows:

$$A_3 = \frac{W_{compre}}{\rho_3 * V_3} \tag{36}$$

$$H_{diff} = \frac{A_3}{\pi*(D_{odif}+D_{idif})/2} \tag{37}$$

By recommended data, for good combustion and low pressure loss, the velocity in the passage V_p and the velocity in the dome V_d should be within 35-60m/s and 7-12m/s respectively. Here 35m/s and 8m/s are chosen, and then get the dome section area A_d and passage area A_p can be calculated as follows:

$$A_d = \frac{W_{c\_d}}{\rho_3 * V_d} \tag{38}$$

$$A_p = \frac{W_{c\_passage}}{\rho_3 * V_p} \tag{39}$$

The combustor reference area (A_{ref}) is the sum of both areas:

$$A_{ref} = A_d + A_p \tag{40}$$

The reference diameter (D_{ref}) is:

$$D_{ref} = \frac{A_{ref}}{\pi * (D_{odif} + D_{idif})/2} \tag{41}$$

The dome height (H_d) and overall passage height (H_p) can be calculated:

$$H_d = \frac{A_d}{\pi * (D_{odif} + D_{idif})/2} \tag{42}$$

$$H_p = D_{ref} - H_d \tag{43}$$

Singh [8] has mentioned that for modern civil engine combustors, the liner length to depth ratio is about 2. Therefore, the length of the liner can be calculated as follows:

$$L_l = 2 * H_d = 0.1805\text{m} \tag{44}$$

Verification by using θ parameter method:

$$\theta = P_3^{1.75} A_{ref} {D_{ref}}^{0.75} \left[\frac{\exp\left(\frac{T_3}{300}\right)}{W_3}\right] \tag{45}$$

The value of theta parameter if is the range of values which are multiples of 10 [Khandelwal, 2011 (a)] then the combustion efficiency is said to be above 99.9% [Khandelwal, 2011 (a)].

A.2.5 COMPONENT DESIGN

DIFFUSER

The dump diffuser design is chosen here, since dump design offers the best compromise between pressure losses and length of the diffuser. Although diffuser designs such as vortex control diffusers or hybrid diffusers are known to have better performance comparatively, the dump diffuser design is the most proven of all and offers better design predictability unlike vortex control designs. A faired diffuser design would give a proven diffuser but it is found inferior in the performance arena. Hence a dump diffuser design is chosen.

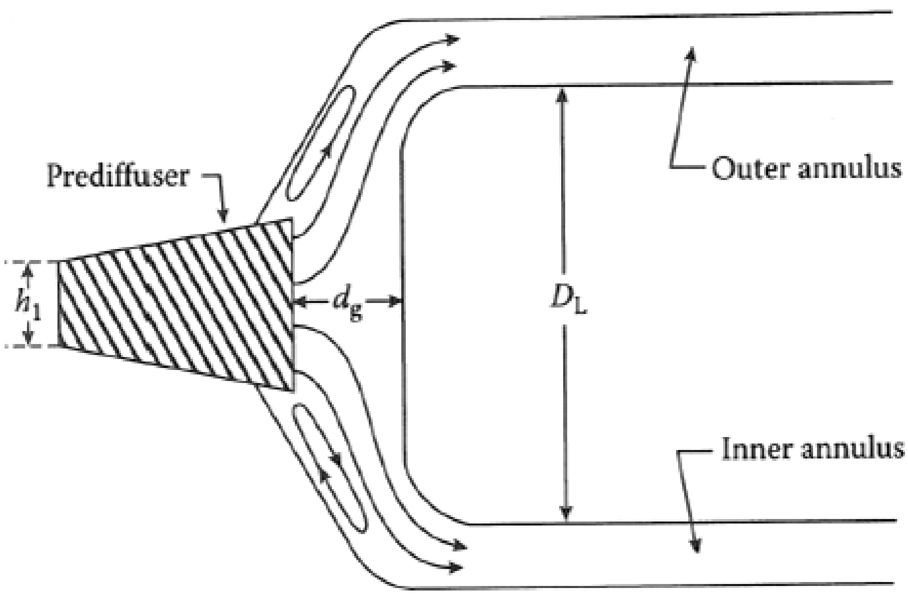


Figure A-16 Dump diffuser [Mohammad, 2009].

According to Mohammad and Jeng [2009], the design of pre-diffuser can be as follows:

i. Area Ratio (AR) varies from 1.4-3; in the model combustor, an area ratio of 1.8 is selected
ii. Calculate the length to width ratio (LW):

$$\mathbf{LW} = \left(\frac{\mathbf{AR}}{\mathbf{1.044}}\right)^{\mathbf{1/0.38859}} - \mathbf{0.26}, \quad (\mathbf{AR} < \mathbf{2}) \tag{46}$$

$$\mathbf{LW} = \mathbf{44.8535}/(\mathbf{1} + \mathbf{2165.67} * \mathbf{e}^{-\mathbf{2.8225*AR}}), (\mathbf{AR} > \mathbf{2}) \tag{47}$$

iii. Calculate the ideal pressure recovery coefficient (C_{pi}):

$$\mathbf{C_{pi}} = \mathbf{1} - (\mathbf{1}/\mathbf{AR^2}) \tag{48}$$

iv. Calculate the actual pressure recovery coefficient (C_p):

$$\mathbf{C_p} = -\mathbf{0.918} * \mathbf{AR^{-0.5}}/\mathbf{LW} + \mathbf{0.677} * \mathbf{AR^{-2}}/\mathbf{LW^2} + \mathbf{0.74} \tag{49}$$

v. Calculate the pressure loss in the aerodynamic diffuser as a fraction of its inlet total pressure:

$$\boldsymbol{\varepsilon_{pd}} = \left(\mathbf{C_{pi}} - \mathbf{C_p}\right)\left(\mathbf{1} - \left(\mathbf{1} + \frac{\boldsymbol{\gamma}-\mathbf{1}}{\mathbf{2}} * \mathbf{M_{3.0}}^2\right)^{\frac{-\gamma}{\gamma-1}}\right) \quad (50)$$

vi. Calculate the pressure loss in the dump diffuser as a fraction of its total inlet pressure:

$$\boldsymbol{\varepsilon_{dd}} = \left(\mathbf{1} - \left(\frac{\mathbf{A_{3.0}}}{\mathbf{A_{ref}}} * \mathbf{AR}\right)^2\right)\left(\mathbf{1} - \left(\mathbf{1} + \frac{\boldsymbol{\gamma}-\mathbf{1}}{\mathbf{2}} * \left(\frac{\mathbf{M_{3.1}}}{\mathbf{AR}}\right)^2\right)^{\frac{-\gamma}{\gamma-1}}\right) \quad (51)$$

$$\mathbf{M_{3.1}} = \mathbf{M_3}/\mathbf{AR} \quad (52)$$

vii. Calculate total pressure loss in the diffuser section:

$$\boldsymbol{\varepsilon_t} = \mathbf{1} - \left(\mathbf{1} - \boldsymbol{\varepsilon_{pd}}\right) * (\mathbf{1} - \boldsymbol{\varepsilon_{dd}}) \quad (53)$$

The least value for this is found to be 0.013 for the model combustor

viii. Check θ and it should be within 4° to 8°:

$$\tan\boldsymbol{\theta} = (\mathbf{AR} - \mathbf{1})/(\mathbf{2LW}) = 5.1^{o} \quad (54)$$

ix. Check the length to width ratio LW. It should be in the prescribed range. (Consider weight etc.)

x. Plot ε_t vs. AR and choose the value of AR that corresponds to the minimum ε_t.

After several iterations, AR=1.8 was selected, and got:

LW=3.8

$$\varepsilon_t = 0.017$$

$$\theta = 5.1°$$

And the pre-diffuser length $L_{diff} = H_{diff} * LW = 0.046\text{m}$

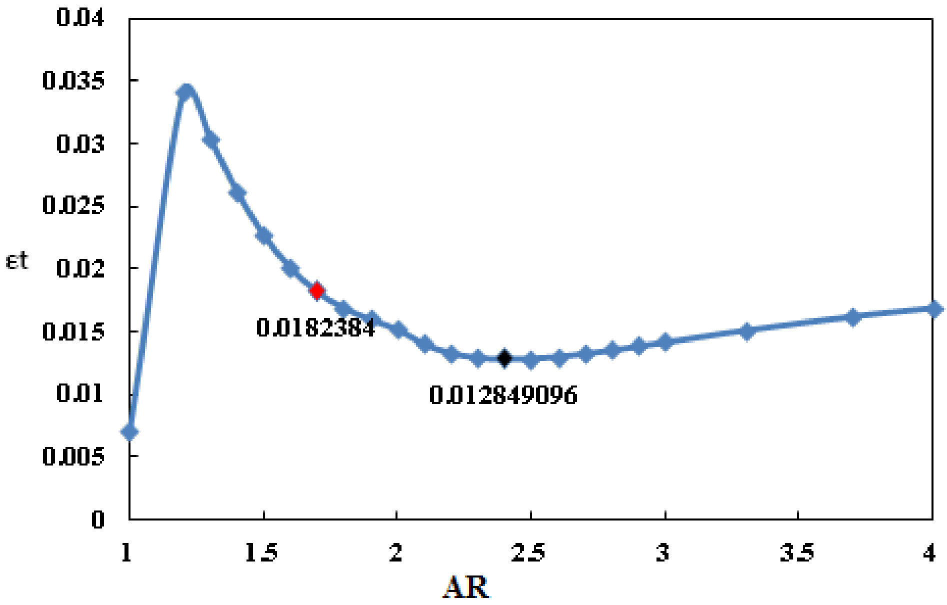


Figure A-17 Graph showing the least total pressure loss (in black) and selected pressure loss for model combustor (in red) with respect to the area ratio.

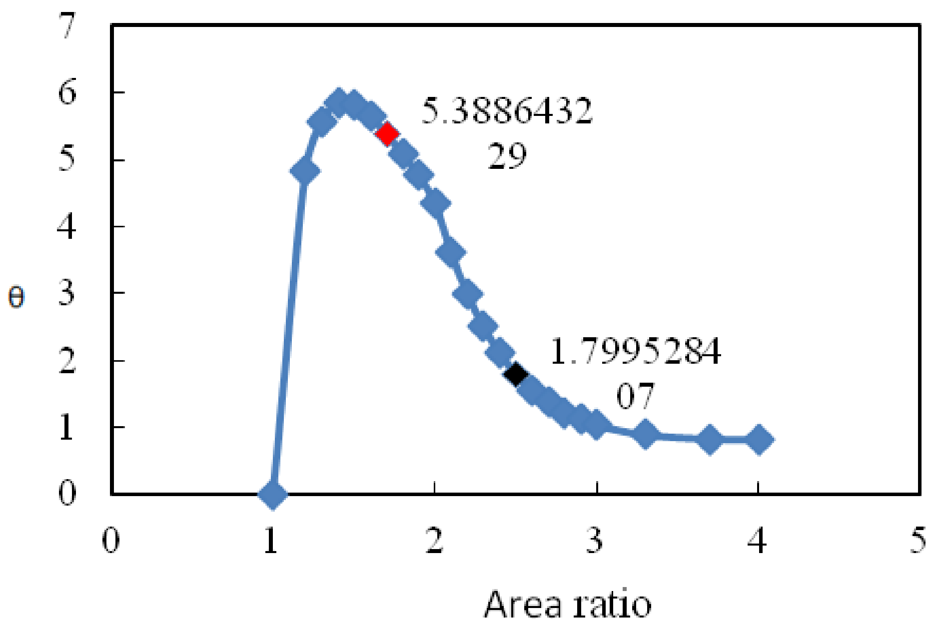


Figure A-18 Graph showing value of *θ* corresponding to the least total pressure loss (in black) and selected pressure loss for model combustor (in red) with respect to the area ratio.

The dump gap ratio is the ratio of the length of dump gap to the height of diffuser inlet. According to Bryn Jones [9], the dump gap ratio should be about 09-1.43 to minimize the pressure loss. Therefore the gap should be:

$$1 \times H_{diff} = 0.0122m \quad (55)$$

SWIRLER

Many designs exist for modern combustor swirlers. There are radial and axial flow swirlers with single or multi pass designs already in use. The authors have considered the most simple of the available designs as this component can be chosen based on options in a combustor design code. The simplest design offers flexibility in computer coding and adaptation with least number of iterations for the whole process. Although it was envisaged to put multiple types of swirlers and injectors in to this work, it was not completely pursued due to time restrictions and limited coding knowledge. Although swirlers have great impact on heat transfer and chemical kinetics of the combustor, the preliminary design as proposed in this paper does not go in to such depths of combustor design.

The typical parameters for axial swirler design include: the vane angleθ, should be 30~60º; the vane thickness should be 0.7~1.5mm and the number of vanes should be 8~16. The definitions of these parameters are shown in Figure 4-9. For radial swirler, the design rules are almost the same as those of axial swirlers [Mellor, 1990; Jeng, 2004].

The swirler design developed in this work is same for both the main and the pilot stage. The design of swirler needs many empirical data or they must be assumed. The assumptions made here are, swirler constant (Ksw), Swirler vane angle, Swirler air flow, Vane thickness (Vt), Number of vanes (Nv), Outer swirler diameter (Dsw), Hub diameter (Dhub). With the total pressure drop across the swirler as input, the calculations for the swirler design are done as explained below:

Frontal area of the swirler:

$$A_{sw} = \frac{\pi}{4} \times \left({D_{tip}}^2 - {D_{hub}}^2\right) - 0.5 n_v v_t \times (D_{tip} - D_{hub}) \quad (56)$$

Swirler number:

$$SN = \frac{2}{3}\left\{\frac{\left(1-\frac{D_{hub}}{D_{tip}}\right)^3}{\left(1-\frac{D_{hub}}{D_{tip}}\right)^2}\right\} \times \tan\theta_v \quad (57)$$

And the vane spacing (Vs) is estimated using Eq.2.11. Hence, the vanes chord (Vch) is determined as one to two times of vane spacing.

$$V_s = \frac{\pi \times (D_{tip} + D_{hub})}{2 \times n_v} \quad (58)$$

Fuel injector

Once the combustion chamber sizing is done, the number of fuel injectors are chosen based on the equation:

$$\mathrm{N_n} = \frac{\pi * (\mathrm{D_{odif}} + \mathrm{D_{idif}})/2}{\mathrm{H_d}} \approx 46 \quad (59)$$

The fuel injector design is directly taken from the already existing literature by our group [Khandelwal, 2011 (a)].

Casing & Liner

This section in the coding is created for the detailed design of the casing and the liner. The gap between the casing and liner is found as passage area. The assumptions made here are ratio of dome cooling flow rate through which mp is found and dome passage velocity, used to find the passage area. The step by step procedure followed in this section is discussed with equations below.

$$A\mathrm{P} = \frac{m\mathrm{P}}{\rho 3 V\mathrm{P}} \quad (60)$$

$$A\mathrm{ref} = A\mathrm{d} + A\mathrm{P} \quad (61)$$

$$V\mathrm{ref} = \frac{m\mathrm{c}}{\rho 3 A\mathrm{ref}} \quad (62)$$

$$Hd = \frac{Ad}{2\pi r4} \tag{63}$$

$$Hp = \frac{Aref}{((2\pi r4) - Hd)} \tag{64}$$

Dilution & Cooling

The dilution zone is use to achieve a desired temperature distribution required by turbine components. The quality of temperature distribution is generally measured by “pattern factor (PF)”. The pattern factor is expressed as:

$$PF = \frac{T_{\max} - T_{t4}}{T_{t4} - T_{t3}} \tag{65}$$

The nozzle guide vanes in a gas turbine are expected to withstand the maximum temperature at the combustor outlet plane. This maximum temperature effects directly on the life of the guide vanes. For rotor blades, they withstand the circumferentially averaged temperature at each radius due to rotation. Generally, the arithmetic mean method is used to calculate the circumferential temperature. Another expression is defined to describe the quality of temperature distribution.

$$PF = \frac{T_{mr} - T_{t4}}{T_{t4} - T_{t3}} \tag{66}$$

In general, 20~40 percentage (%) of total combustor inlet air is used for dilution. The dilution air available for modern high temperature rise combustor will be reduced further because more air is used for combustion and cooling.

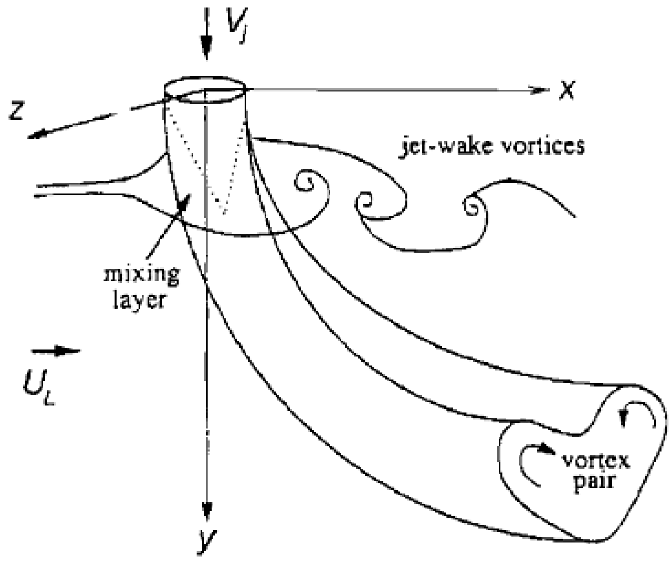


Figure A-19 Jet from a hole into a cross flow [Mellor, 1990]

The dilution air is generally admitted into the liner to mix with the combustion products through dilution holes. shows the typical jet discharge from a reservoir to a cross stream. The injection blocks the mainstream flow. This blockage effect will produce a pressure differential which results in deformation of the jet. The jet penetration of dilution air has significant effect on pattern factors at the combustor outlet plane.

One of the problems in the design of the dilution holes is that the designer is given two options or methods to choose from while both result in slightly varying dimensions. The dilution hole sizing can be designed by two different methods, NASA design method and Cranfield method. . The authors have chosen Cranfield method which emphasises on hole size over the NASA method which focuses mainly on the hole spacing. Both the methods are proven ad also have been found to produce different results. Although there are no particular advantages of choosing one over another, the Cranfield method, as stated by Lefebvre and Ballal [2010] takes in to account the adverse effects on jet penetration and mixing caused by aerodynamic blockages created by adjacent holes.

$$d_{\mathrm{h}} = \frac{d_{\mathrm{j}}}{\sqrt{c_{\mathrm{dd}}}} \quad (67)$$

$$U_{\mathrm{ji}} = \frac{2\Delta P_{\mathrm{l}}}{\sqrt{\rho_3}} \tag{68}$$

$$n = 4N_{\mathrm{f}} \tag{69}$$

$$nd_j^{\ 2} = 15.25 \times m_{\mathrm{j}} \times \left(\frac{P_3 \Delta P_{\mathrm{l}}}{T_3}\right)^{-0.5} \tag{70}$$

Cooling system design for the liner is very important since the liner is expected to contain all the hot combustion gases within it. There are many types of cooling methods that exist in the contemporary combustor designs. The selection of cooling scheme for liner cooling is related to combustor operating parameters, life requirement, manufacture cost and other factors. Conventional film cooling scheme is chosen as the cooling scheme of liners. Figure A-20 shows a typical film cooling configuration. Cooling air flows through slots to produce a thin film at the surface of liners. An ideal film is to flow along the liner wall, isolating high temperature reaction mixture from the liner wall surface.

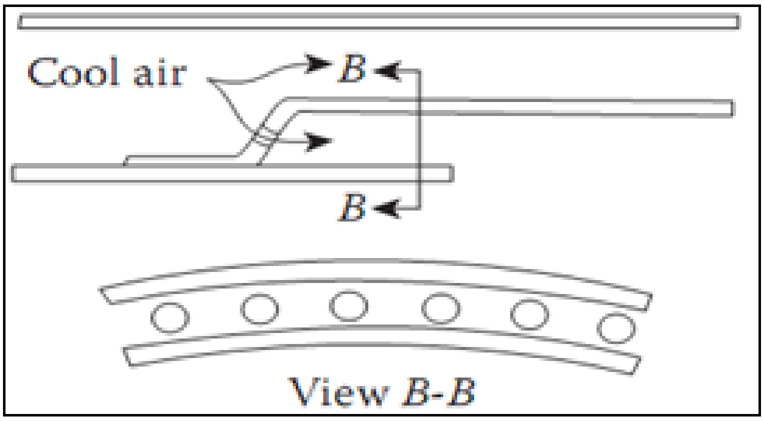


Figure A-20 Film cooling [Lefebvre, 2010]

The impingement cooling scheme is chosen based on simplicity and ease of approach. After the cooling scheme is selected, the next step is to calculate the cooling air flow rate.

In the initial design, the cooling air distribution is assumed. The design parameters of cooling slots include the slot length, the diameter of discrete cooling holes, and the number of cooling holes, etc. The discharge coefficient of cooling hole is assumed to be 0.8. The total effective areas of holes can be estimated:

$$A_{\text{Chole}} = \frac{m_{\text{cooling}}}{C_{\text{dc}} \times \sqrt{2\rho_3 \Delta P_{\text{l}}}} \tag{71}$$

$$D_{\text{Chole}} = \sqrt{\frac{4 \times A_{\text{Chole}}}{\pi(N_{\text{co}} + N_{\text{ci}})}} \tag{72}$$

Mellor suggested that the maximum diameter of discrete holes is about 2.5mm and the minimum distance between two adjacent holes is about 1.25mm. In practice, the design of cooling film is related to a parameter, MIX_N which is defined as the following expression [Gupta, 1984]. The value of MIX_N should be lower than 0.5 [Gupta, 1984]. The model combustor is calculated to have MIX_N of 0.177.

$$MIX_{\text{n}} = \left(\frac{BH_{\text{ou}}l}{D_{\text{Chole}} \times H_{\text{in}} \times L}\right) \tag{73}$$

Where,

B = Axial width of vane passage

H_{ou} = slot height

l = length of jet prior to impingement

DChole = Diameter of cooling holes

Hin= internal height of mixing chamber

L = internal flow path length

MIXn = monitored parameter

A.2.6 Performance Analysis

Efficiency at Design Point:

As we mentioned above, by using the θ parameter method, the efficiency at design point can be estimated to be 99.99%.

Overall Pressure Loss

The total pressure in the combustor can be expressed as follows:

$$\Delta P=\varepsilon_t+\Delta P_l+\Delta P_{comb} \quad (74)$$

where:

ε_t – Pressure loss in the diffuser

ΔP_l – Pressure loss in the liner

ΔP_{comb} – Pressure loss cause by combustion

As we calculated before, $\varepsilon_t=1.9\%$, and according to Lefebvre and Ballal [Lefebvre, 2010] we assume $\Delta P_l=3\%$ and $\Delta P_{comb}=0.5\%$, then,

$\Delta P=5.191\%$

That is within the limit of 6%.

Prediction of NOx from empirical correlations

Many empirical and semi-empirical models are widely used for correlating the experimental data on pollutant emitted and some relevant parameters of combustors. The parameters thus correlated include combustor dimensions, design features, operating conditions; fuel type and fuel spray quality. Hence, the empirical models can play an important role in the design and development of low emission combustor as summarized by Lefebvre [Lefebvre, 2010]. Lefebvre's assumptions follow that the emissions are mainly dependent on three terms; mean residence time in the combustion zone, chemical reaction rates and mixing rate. Hence, expressions for these three parameters are divided in terms of combustor size, liner pressure drop, air flow proportion and operating conditions such as inlet pressure, inlet temperature and air mass flow.

Lefebvre [5] provided an equation to predict the NOx emissions as follows:

$$NO_x=9\times 10^{-8}\times P^{1.25}\times V_c\times \exp(0.01T_{st})/((m_A) T_{pz}) \text{ g/kg of fuel.}$$

For estimating the NOx formation, the values of T_{st} and T_{pz} are assumed to be 1900K and 1700K respectively. This is well below the temperatures experienced in conventional combustors due to the fact that the primary zone in this combustor is expected to operate at an equivalence ratio of 0.6-0.8. The result is NOx emission of the order 50-60 ppm.

The overall design method developed for axial staging does not differ much in comparison to the parallel staged dual annular combustor. However, changes are made in the detailed design of components. The axial staged dual annual combustor modelling too is coded in to excel to aid the user for quick changes and results. The important changes made are the determination of axial spacing between the stages, the consideration of inlet dome cross section area for a single injector. An important assumption on axial spacing between the stages is made based on the available data and discussion with the industrial experts. The spacing is assumed to be the length of the primary zone length of the pilot combustor. The pilot zone is assumed to be operating fuel rich to facilitate easy ignition, good combustion stability and a reliable altitude relight.

Much of the design methodology for axially staged dual annular combustor remains similar to that of the radial staged dual annular combustor. The important change is observed only in the design of casing and liner section of the combustor. Due to axial staging, the combustor length increases and the important assumption is made for the determination of axial stage spacing. As discussed earlier, the spacing is taken as the primary zone length of the pilot burner. This results in staging the design method itself. The liner two as it is termed here is the main burner liner.

A.2.7 Comparison in series and parallel staging

There are advantages and disadvantages of both type of staging when compared with each other. The radial staging results in a shorter, lighter, mechanically robust and simpler design with modest control mechanisms, while the axial staging of combustion chamber results in a more efficient, less polluting and stable design.

The radial staged combustor although shorter, has a bigger cross sectional area hence results in engine of bigger casing diameter. This is evident from the design of the model combustor where, the radial staged combustor has a dome height of 0.078m compared to a dome height of 0.039m for axially staged combustor. At the same time the total length of the radial staged combustor is found to be 0.368m while the axially staged combustor will have a length of 0.415m. Due to longer liner, the surface area exposed to the hot gases in an axially staged combustor is more than the radially staged combustor. This results in requirement of more cooling flow for the combustor liner.

In terms of emission and efficiency, axial staging does have some advantages over radial staging. In an axial staged combustor since the main stage is downstream of the pilot stage, ignition of the main stage directly from pilot stage is both rapid and reliable. Also the hot gas flow in to the main stage ensures high combustion efficiency even at lower equivalence ratios. Rapid combustion means the design can be tuned to have lower residence time for efflux gases. Since the formation of thermal NO_x is directly proportional to the residence time of gases at higher temperature, the reduction in residence time has a direct effect in reducing the NO_x emission of the combustor. This point is also proven by the developed model combustor. The NO_x prediction model used in this work is very crude and not accurate. However, the gradient or the change in the emission levels can be considered as useful results. The NO_x emission from the model radially staged combustor is found to be 55ppm while the axial combustor is predicted to emit 45ppm of NO_x. Although the exact values are not to be taken for the comparison, one can clearly make out that the axially stages combustor has lower emission prediction using the same method applied to predict emission of radially staged combustor.

Table 14. Input parameters

PARTS	INPUT PARAMETERS	
	NON GEOMETRICAL PARAMETERS	GEOMETRICAL PARAMETERS
PRE-DIFFUSER	Fuel flow rate	Inlet section diameter of diffuser
	T3(K)	liner length/dome height
	M3	
	P3(Kpa)	
SWIRLER	Pressure loss in liner(Pl) %	Vane chord
	Combustion pressure loss(Pcomb) %	Diameter of the injection holes

Tables 15 and 16 give the summaries of the input and output parameters obtained from the combustor modeling code.

Table 15 Output parameters

PARTS	OUTPUT PARAMETERS	
	NON GEOMETRICAL PARAMETERS	GEOMETRICAL PARAMETERS
PRE-DIFFUSER	FAR (Kg/S)	Inlet area of diffuser (Adi)
	Combustion air flow rate	Inlet height of the diffuser(Hdi)
	t3	R3
	p3	
	rho3	
	v3	
DIFFUSER	Ideal pressure recovery coefficient (Cpi)	Length of diffuser
	Actual pressure recovery coefficient (Cp)	
	Pressure loss in the aerodynamic diffuser	Area Ratio (AR)
	Pressure loss in the dump diffuser	
	Total pressure loss in the diffuser	
SWIRLER		Frontal area of swirler (Asw)
	Swirler number (Sn)	Vane spacing (Vs)
CASING AND LINER	Air flow in dome (md)	Dome area (Ad)
	Reference area (Aref)	Passage area (Ap)
	Reference velocity (Vref)	Height of dome (Hd)
		Height of passage (Hp)
DILUTION AND COOLING	Jet velocity at dilution zone(Uj)	Area of cooling hole (Ahole)
	MIXn	Diameter of holes
		Actual diameter of dilution holes (dh)
		Effective diameter of dilution holes(dj)
		Number of dilution holes (n)
		number of cooling holes inner liner
		number of cooling holes outer liner

A.2.8 Conclusion

A detailed methodology has been proposed for gas turbine combustor with detailed sheet for each of the parts in a combustor. The diffuser design is based on well tried and tested area ratio method. The length of the diffuser is selected considering the pressure loss factor and the divergent angle. Although minimum loss approach yield better performance, the length of such diffusers will be impractical hence a compromised value is chosen. The radial staging assumes an equivalence ratio well below 1 (stoichiometric combustion mixture) for low emission purposes. This results in a bigger frontal cross sectional area. Since the volume of the combustor is kept at the optimum level an increase in width of the combustor results in the reduction of the length. The distance between the pilot and the main combustor is assumed to be the length of the primary zone of the pilot burner. The main conclusion is that the dimensions and preliminary results of model combustors obtained from the developed design methodology are within the feasibility limits. The comparison between the radially staged and the axially staged combustor has yielded the predicted results such as lower NOx prediction for the latter and shorter combustor length for the former. A performance prediction of the combustors is also made which show the combustion efficiency of 0.999 with a predicted total pressure loss of 5.2% and an approximate NOx emission of 55ppm for the radially staged combustor and about 45ppm for the axially staged combustor. However, it must be noted that the excel sheet has its own limitations. An important limitation being, only one combustor can be designed at a time and that the code relies heavily on the available empirical data hence the developed code is not very robust.

A.3 REVERSE-FLOW COMBUSTOR

A.3.1 Abstract

Modern advanced engines are expected to operate with higher combustor temperature rise and lower emissions. These development trends result in more combustor design difficulties. High temperature rise requires more air for complete combustion, hence reducing the amount of cooling air. Emissions consist of CO_2, UHC, NO_x, smoke and water vapor. CO_2 is an unavoidable emission of combustion reaction, whereas emissions of NO_x mainly depend on temperature. A lean primary zone design is required to achieve low NOx emission. Reverse flow combustors have more difficulties because of the presence of turn section which does not exist in other combustor configurations.

There are many studies in public domain which talk about design of combustors. But none of them gives detailed guide on designing reverse flow combustor. The objective of this paper is to provide a reverse flow combustor design procedure, with emissions and performance analysis. The combustor designed in this study is expected to be used in advanced helicopter engine. Substantial amount of literature is available on conventional combustor designs which mainly include empirical and semi-empirical models plus experiment test methods. All these combustor design methods focus on the direct flow combustor. In this study, a reverse flow combustor design methodology is proposed. The design procedure includes the combustor sizing, fuel injector design, swirl cup design, air distribution along the liner, primary hole design, dilution zone design and the cooling system design. Final dimensions are also shown in a figure, which have been validated with one of the present combustor design. After finalizing the design of the combustor, overall performance has been evaluated using empirical correlations and equations.

A.3.2 INTRODUCTION

The aero-engine combustor design is a challenging task because it involves very complicated flow phenomena and many requirements which are always needed to be balanced. The actual combustor design is an iterative process. In this paper, a reverse flow combustor preliminary design procedure and some key component preliminary design procedures are presented. Based on some empirical correlations and preliminary design dimensions, the overall performance is evaluated. A new combustor design depends heavily on the previous experience [Leishman, 2000]. Some combustor design rules are generally related empirical or semi-empirical correlations. A large number of combustor design approaches have been presented [Mellor, 1990; Mellor, 1990 (a); Mohammad, 2009; Danis, 1997].

However, most combustor design methods are focused on large engine combustors. For small gas turbine combustors design, the application of these design approaches suffers from size-related effects [Leyes, 1999]. Therefore, some techniques used in large combustors cannot directly be scaled to small engine combustor design [Demetri, 1980]. This combustor design is only based on some empirical or semi-empirical correlations and recommended data.

In this study, a reverse flow combustor design methodology is presented. The design procedure includes the combustor sizing, fuel injector design, swirl cup design, air distribution along the liner, primary hole design, dilution zone design and the cooling system design. Final dimensions are also shown in a figure, which have been validated with one of the present combustor design.

A.3.3 Design procedure

The temperatures and pressures at combustor inlet and outlet are obtained by overall engine operating parameters at design point. This data is used to initiate design process.

A.3.4 Combustor type and size

For small engines, reverse flow combustors are widely used due to the presence of centrifugal compressor which makes the engine shorter. The combustor size can be calculated by many methods like θ parameter approach, velocity method, pressure loss, etc. In this study reference velocity method is used to determine combustor size. Due to the large radial space between the compressor outlet and the turbine inlet, reference velocity in reverse flow combustors is much lower than the reference velocity in axial flow combustors. The primary zone outlet velocity, primary air mass flow, outer and inner annulus passages velocity are assumed according to the literature.

A.3.5 Air distribution along the liner

Air mass flow at different combustor positions is allocated according to the purposes. The air distribution includes the dome cooling air mass flow, the primary holes air mass flow, the dilution holes mass flow and cooling air mass flow. The mass flow in primary zone is decided by the assumption of the equivalence ratio in primary zone. Based on the low emission consideration, equivalence ratio is kept lean.

Component design

Due to the low compressor exit Mach number, the combustor diffuser is not necessary in reverse flow combustor configuration, hence not considered in this study.

3.4 The air blast fuel injector

The air blast fuel injector type is chosen due to its low emission characteristics. Air blast atomizer can effectively reduce the smoke emission. The design procedure of fuel injector is provided according to some empirical methods and recommended data [4-5], the air blast atomizer has emissions as follows:

UHC (g/kg) ≈0, CO (g/kg)= 1.6, NO_x (g/kg)= 13, Smoke (number)= 15.

The number of fuel injectors to be used in a combustor is calculated by the following relation,

$$N_{f,opt} = \frac{2 \times \pi \times R_c}{H_{dome}} \tag{75}$$

<u>Design Procedure</u>

(1) The fuel flow rate in each atomizer,

$$m_{f,pn} = \frac{m_f}{N_f} \tag{76}$$

The total air flow rate for each atomizer, The ratio of air to fuel by mass should be about 3 [Lefebvre, 1989].

$$m_{a,pn} = 3 \times m_{f,pn} \tag{77}$$

(2) The total atomization air,

$$m_{a,ato} = N_f \times m_{a,pn} \tag{78}$$

So, the fraction of air for atomization is:

$$\frac{m_{a,ato}}{m_{a,3}} \tag{79}$$

(3) The air flow rates in inner passage and in outer passage are assumed to be equal.

$$m_{a,inner\_pn} = m_{a,outer\_pn} = 0.5 \times m_{a,pn} \tag{80}$$

(4) The pressure drop in air blast atomizer,

The pressure drop in air blast atomizer is assumed to equal the overall total pressure loss in the combustor because of the absence of the diffuser and the low air velocity.

$$\Delta P_{pn} = P_{t3} - P_{t4} \tag{81}$$

(5) The effective area of inner passage and outer passage:

$$A_{inner\_pn,eff} = A_{outer\_pn,eff} = \frac{m_{a,inner\_pn}}{\sqrt[2]{2 \times \rho_3 \times \Delta P_{fn}}} \tag{82}$$

(6) The discharge of coefficients of both inner passage and outer passage are assumed to be 0.7. The definition of discharge of coefficient and its influencing factor will be described later.

$$C_{d,fn} = 0.7 \tag{83}$$

(7) The physical minimum flow area of both inner and outer air passage.

$$C = A_{outer\_pn,phy} = \frac{A_{inner,pn,eff}}{C_{d,fn}} \tag{84}$$

The main configuration of the air blast atomizer is shown in Figure A-21. An air blast atomizer designed by Lefebvre [Lefebvre, 1989; Gupta et al., 1984]. is used for reference. The effective flow areas for inner and outer air passages are assumed to be located at individual swirler passage. At the outlet plane, the outer and inner air passages have the equal area.

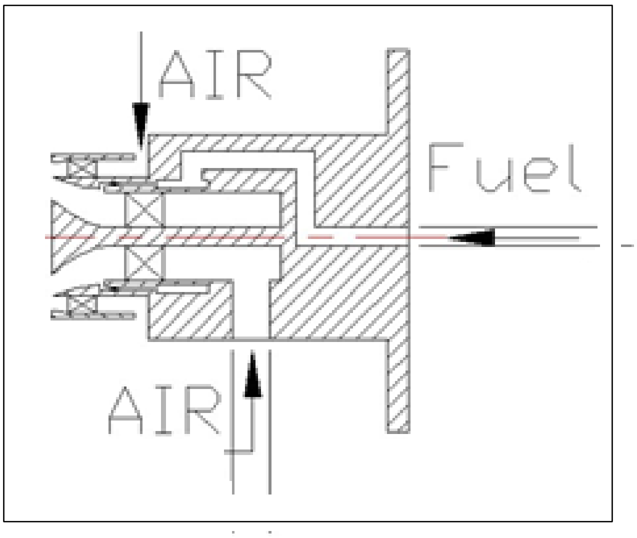


Figure A-21 Schematic diagram of air blast atomizer [Khandelwal, 2011(a)]

Swirler Design

The swirl cup is designed to provide rapid mixing of fuel droplet and air, and for good atomization in the primary zone [Mellor, 1990]. This reduces the liner length, which can be significantly beneficial to the cooling design due to the less liner surface area. In this combustor, a jets-radial swirl is applied.

The swirl number reflects the degree of swirling flow and is defined as:

$$S_N = \frac{G_\theta}{G_x \times R_{sw}} \tag{85}$$

The swirl number has a direct effect on the recirculation zone. A recirculation flow will be set up when the swirl number is higher than 0.6 [Gupta et al., 1984].

<u>Design Procedure</u>

The flow rates in primary swirler and secondary swirler are assumed firstly.

It is adequate to assume the equivalence ratio at the swirler outlet plane is no more than 2.0 [Mellor, 1990], namely:

$$\frac{m_f}{(m_{a,ato} + m_{a,sw})} \times \frac{1}{FA_{th}} < 2 \tag{86}$$

$$m_{a,ato} + m_{a,sw} > \frac{m_f}{2 \times FA_{th}} \tag{87}$$

The minimum flow rate of atomization air plus swirling air should be 20.2% of total air flow.

In this case, the flow rates in the primary swirler and secondary swirler are assumed to be X% and Y% of the total mass flow, respectively.

I. The primary swirler design procedure

(1) The calculation of the flow rate:

$$m_{a,sw\_pri} = m_c \times X\% \tag{88}$$

(2) The number of swirlers:

The number of swirlers is equal to the number of fuel injectors.

$$N_{sw\_pri} = N_f \tag{89}$$

(3) The air flow rate in each swirler:

$$m_{a,sw\_pri\_pn} = \frac{m_{a,sw\_pri}}{N_{sw\_pri}} \tag{90}$$

(4) The pressure drop in the swirler:

The pressure drop across the primary swirler is assumed to equal the overall pressure loss in the combustor.

$$\Delta P_{sw\_pri} = P_{t3} - P_{t4} \tag{91}$$

(5) The total effective area of each primary swirler:

$$A_{sw\_pri,eff} = \frac{m_{a,sw\_pri}}{\sqrt[2]{2 \times \rho_3 \times \Delta P_{sw\_pri}}} \tag{92}$$

(6) The discharge of coefficient of both inner passage and outer passage are assumed to be 0.7 [Mellor, 1990].

$$C_{d,sw\_pri} = 0.7 \tag{93}$$

(7) The physical flow area of each primary swirler:

$$A_{sw\_pri,phy} = \frac{A_{sw\_pri,eff}}{C_{d,sw\_pri}} \tag{94}$$

(8) The number of ellipsoidal holes is assumed to be N.

$$N_{sw\_pri\_hole} = N \tag{95}$$

(9) The area of each ellipsoidal hole:

$$A_{sw\_pri\_hole} = 20 / N_{sw\_pri\_hole} \tag{96}$$

The equatorial radii are assumed. The rotational direction of air is clockwise.

II. The secondary swirler design procedure

(1) The calculation of the flow rate

$$m_{sw\_sec} = m_{a,3} \times Y\% \tag{97}$$

(2) The number of secondary swirlers:

The number of secondary swirlers is equal to the number of primary swirlers.

$$N_{sw\_sec} = N_{sw\_pri} \tag{98}$$

(3) The air flow rate in each swirl:

$$m_{a,sw\_sec} = \frac{m_{a,sw\_sec}}{N_{sw\_sec}} \tag{99}$$

(4) The pressure drop in secondary swirler:

The pressure drop in the primary swirler is assumed to equal the overall pressure loss in the combustor.

$$\Delta P_{sw\_sec} = P_{t3} - P_{t4} \tag{100}$$

(5) The total effective area of each secondary swirler:

$$A_{sw\_sec,eff} = \frac{m_{a,sw\_sec}}{\sqrt[2]{2 \times \rho_3 \times \Delta P_{sw\_sec}}} \tag{101}$$

(6) The discharge of coefficient is assumed to be 0.7 [Mellor, 1990].

$$C_{d,sw\_sec} = 0.7 \tag{102}$$

(7) The swirl physical flow area of each primary swirler:

$$A_{sw\_sec,phy} = \frac{A_{sw\_sec,eff}}{C_{d,sw\_sec}} \tag{103}$$

(8) The number of vanes is assumed to be M.

$$z = M \tag{104}$$

(9) The angle of vanes is assumed to be 60°.

$$\theta_{sw\_vane} = 60° \tag{105}$$

(10) The axial width of passages:

Figure A-22 shows the schematic diagram of the swirl cup. The diverging angle of flare is 90°. The throat area of primary swirler is the total area of N holes. For the secondary swirler which generates a counter-rotating air flow, the effective flow area is located in the outlet passage, because the height of vanes will be very small, hence increasing the cost and the difficulty of cast of swirlers if this area is at the vane passages.

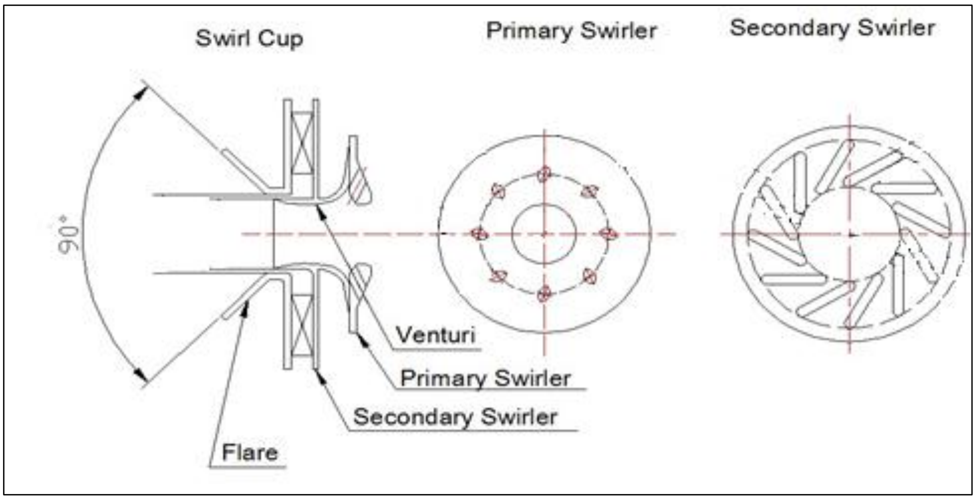


Figure A-22 Schematic diagram of swirl cup [Khandelwal, 2011(a)]

Evaluation of Swirl Number

The swirl number should be higher enough to produce a recirculation zone in primary zone. For the swirl cup, the strong secondary swirling air is used to achieve this function [Jeng et al., 2004]. Therefore, it is necessary to evaluate the swirler number for the secondary swirler.

Beer and Chigier introduced an approach to calculate the swirl number for radial swirlers [Beer and Chigier, 1972]. This approach gives a good approximation although it omits the effects of pressure and Reynolds Number, etc.

The axial flux of swirl momentum is expressed as:

$$G_\theta = \frac{\sigma \times m_{a,sw\_\sec}^2}{\rho_a \times 2 \times \pi \times B} \tag{106}$$

where

$$\sigma = \frac{1}{1 - \dfrac{z \times t}{2 \times \pi \times R_{out,\sec\_sw} \cos\alpha}} \times \frac{\tan\theta}{1 + \tan\theta \times \tan(\pi / z)} \tag{107}$$

Hence,

$$G_\theta = \frac{\sigma \times m_{a,sw\_\sec}^2}{\rho_a \times 2 \times \pi \times B} \tag{108}$$

The axial flux of swirl momentum is:

$$G_x = \int_{R_{inner\_ann,sec\_sw}}^{R_{outer\_ann,sec\_sw}} 2 \times \pi \times \rho_3 \times V^2 \times rdr \quad (109)$$

$$= V \times \rho_3 \times A_{sw\_sec,phy} \times V \quad (110)$$

In the above equation, the velocity distribution in the secondary swirler outlet passage is assumed to be uniform. According to the mass flow and physical area, the average velocity is obtained. Hence, the swirl number is:

$$S_N = \frac{G_\theta}{G_x \times R_s} \quad (111)$$

3.3.3 Cooling System Design

The main difficulty in reverse flow combustor design is the cooling system design. The liners suffer complex loads, including aerodynamic loads, thermal loads, and mechanical loads, etc. Substantial amount of air is used to complete the combustion and only a part of the air is required to cool the turn section. In this design, film cooling technology and effusion cooling technology are used to cool the liner and turn section respectively.

(1) Film cooling

Figure 3 shows a typical film cooling configuration [Lefebvre, 1989; Gupta et al., 1984]. Cooling air flows through slots to produce a thin film at the surface of liners. An ideal film is to flow along the liner wall, isolating high temperature reaction mixture from the liner wall surface.

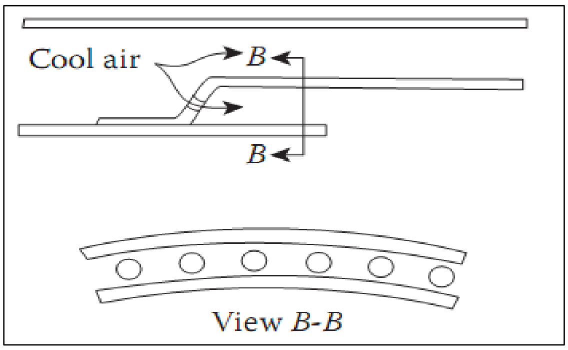


Figure A-23 Film cooling [Khandelwal, 2011(a)]

Dome Cooling Design

The impingement plus film cooling scheme is chosen to cool the liner dome. A heat shield plate is designed between the dome and the swirler outer flare. The cooling air issued from discrete small holes on the dome first impinges to the heat shield plate, and then form a cooling film to cool the dome. The configuration insulates effectively radiation heat to the dome because of the existence of the heat shield plate, hence reducing the stresses of the dome support structure and improving the liner life [Mellor, 1990].

According to the area of the heat shield plate, the cooling air flow, and the percentage of dome cooling air can be evaluated:

$$m_{a,dome} = 0.5 \times A_{shiled} \times \frac{P_3}{101325} \quad (112)$$

The typical recommended value is between 10~15%.

The diameter of the discrete holes is assumed and is chosen to be higher than the lowest limit of 0.4 mm presented by Lefebvre [Lefebvre, 1989; Gupta et al., 1984]. So, if the discharge coefficient of holes is assumed, the physical cooling area can be calculated:

$$A_{dome,phy} = \frac{m_{a,dome}}{C_{d,d} \times \sqrt[2]{2 \times \rho_3 \times \Delta P_{liner}}} \quad (113)$$

The number of discrete holes can be calculated:

$$N_{dome\_hole} = 4 \times \frac{A_{dome,phy}}{\pi \times d_{d,hole}^2} \quad (114)$$

Liner Cooling Design

(Ⅰ) Selection of cooling scheme

The selection of cooling scheme for liner cooling is related to combustor operating parameters, life requirement, manufacture cost and other factors. Conventional film cooling scheme is chosen as the cooling scheme of liners.

(Ⅱ) Preliminary cooling flow rate estimate

After the cooling scheme is selected, the next step is to calculate the cooling air flow rate.

In the initial design, the cooling air distribution is assumed.

Detailed design of cooling slots

The design parameters of cooling slots include the slot length, the diameter of discrete cooling holes, and the number of cooling holes, etc. The discharge coefficient of cooling hole is assumed to be 0.8 [Mellor, 1990]. The total effective areas of holes can be estimated:

$$A_{hole,eff} = \frac{m_{a,slot}}{\sqrt[2]{2 \times \rho_3 \times \Delta P_{liner}}} \quad (115)$$

Mellor suggested that the maximum diameter of discrete holes in about 2.5mm and the minimum distance between two adjacent holes is about 1.25mm [Mellor, 1990]. In practice, the design of cooling film is related to a parameter, MIX_N which is defined as the following expression [Mellor, 1990]. The value of MIX_N should be lower than 0.5 [6].

$$MIX_N = \frac{P \times S \times I}{d \times D \times L} \quad (116)$$

Thermal barrier coating

The application of thermal barrier coatings is an attractive way to solve the difficulty of liner cooling. Thermal barrier coatings are refractory materials with low emissivity and low thermal conductivity. Typical thermal barrier coatings can reduce the liner wall temperature about 40~70K [Lefebvre, 1989; Gupta et al., 1984].

Outer and Inner Turn Section Cooling Design

The effusion cooling approach is chosen to cool the inner and outer turn sections.

(1) The surface areas of outer and inner turn section

$$A_{outer\_turn} \quad (117)$$

$$A_{inner\_turn} \quad (118)$$

(2) The total cooling air

$$m_{a,outer\_turn} \quad (119)$$

$$m_{a,inner\_turn} \quad (120)$$

(3) The discharge coefficient of cooling holes is assumed to be 0.7.

$$C_{d,inner\_turn} = C_{d,outer\_turn} = 0.7 \quad (121)$$

(4) The total effective area of inner and outer holes at the turn section are defined as:

$$A_{outer\_turn,eff} = \frac{m_{outer\_turn}}{\sqrt[2]{2 \times \rho_3 \times \Delta \mathrm{P}_{\mathrm{liner}}}} \quad (122)$$

$$A_{inner\_turn,eff} = \frac{m_{inner\_turn}}{\sqrt[2]{2 \times \rho_3 \times \Delta \mathrm{P_{liner}}}} \tag{123}$$

(5) The total physical area of inner and outer holes at the turn section are defined as:

$$A_{outer\_turn,phy} = \frac{A_{outer\_turn,eff}}{C_{d,outer\_turn}} \tag{124}$$

$$A_{inner\_turn,phy} = \frac{A_{inner\_turn,eff}}{C_{d,inner\_turn}} \tag{125}$$

(6) The diameter of holes

The cooling effectiveness will be increased when the diameter of cooling holes is decreased. In practical combustors, the diameter of holes should be no less than 0.5mm when the blocking effect is considered.

$$d_{inner\_turn\_hole} = d_{outer\_turn\_hole} \tag{126}$$

(7) The numbers of holes in outer and inner turn sections:

$$N_{outer\_turn\_hole} = \frac{A_{phy,outer\_turn}}{\frac{1}{4} \times \pi \times d^2_{hole,turn}} \tag{127}$$

$$N_{inner\_turn\_hole} = \frac{A_{phy,inner\_turn}}{\frac{1}{4} \times \pi \times d^2_{hole,turn}} \tag{128}$$

(8) The number of holes per unit area at outer and inner turn section:

$$N_{outer\_turn,unit} = \frac{N_{outer\_turn}}{A_{outer\_turn}} \tag{129}$$

$$N_{inner\_turn,unit} = \frac{N_{outer\_turn}}{A_{outer\_turn}} \tag{130}$$

Primary Hole Design

The primary holes play important roles in the primary zone. For obvious reasons, it is necessary to reduce the length of primary zone. Some analyses indicate that the primary holes determine the axial length of the primary zone [Mellor, 1990].

i. Location of Primary Holes

The primary holes are located downstream of the fuel injectors. The length from the fuel injector outlet to the central line of primary holes equals approximately the line dome height [Mellor, 1990].

ii. Type of Primary Holes

Several types of holes have been successfully used in practical aero engine combustors, including plain holes, plunged holes rectangular slots and elliptical holes, etc. The shape and type of holes have effects not only on the cost, but also the discharge coefficient, and the penetration depth of jets, etc.

The discharge coefficient is defined as:

$$C_d = \frac{m_h}{A_{phy} \times 2 \times \rho_3 \times \sqrt[2]{P_1 - p_j}} \qquad (131)$$

The main factors influencing the discharge coefficient include the shape, size and type of hole, the pressure differential across the liner, etc [Mellor, 1990; Lefebvre, 1989; Gupta et al., 1984]. For plain holes, the discharge of coefficient can be expressed as:

$$C_d = \frac{1.25 \times (K-1)}{\left[4 \times K^2 - K \times (2-\alpha)^2\right]^{0.5}} \qquad (132)$$

In general, the flow through a liner hole is not along with the normal direction of the hole. Figure 4 shows the flow through a hole [Lefebvre, 1989; Gupta et al., 1984]. This jet angle affects directly on the discharge coefficient of holes because it reduces the effective area of the hole.

The initial jet angle is defined as:

$$\sin^2 \theta_j = \frac{C_d}{C_{d,\infty}^2} \qquad (133)$$

$C_{d,\infty}$

The value of

$$C_{d,\infty} = \lim_{K \to \infty} \left(\frac{1.25 \times (K-1)}{[4 \times K^2 - K \times (2-\alpha)^2]^{0.5}} \right) \qquad (134)$$

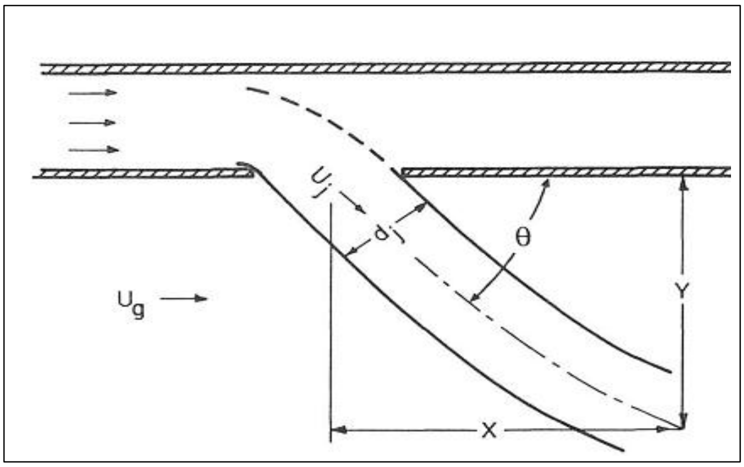


Figure A-24 Flow regimes through holes [Lefebvre, 2010]

iii. Design Procedure

(1) The pressure loss across the liner:

The pressure loss across the liner is assumed to equal to the overall total pressure loss because the velocity in the combustor is relatively low. For the same reason, the static pressure is also assumed to be equal to the total pressure.

$$\Delta P_{liner} = P_{t3} - P_{t4} \tag{135}$$

(2) The jet velocity:

$$V_j = \sqrt[2]{2 \times \frac{\Delta P_{liner}}{\rho_3}} \tag{136}$$

(3) The hole pressure loss factor:

In outer annular passage, the inner and outer pressure loss factors are defined by:

$$K_{outer} = \frac{\frac{1}{2} \times \rho_j \times V_j^2}{\frac{1}{2} \times \rho_j \times V_{pass}^2} \tag{137}$$

$$K_{inner} = \frac{\frac{1}{2} \times \rho_j \times V_j^2}{\frac{1}{2} \times \rho_j \times V_{pass}^2} \tag{138}$$

(4) The air flow rates through outer liner primary holes and inner liner primary holes are assumed to be 6% of total mass flow rate, respectively.

$$m_{a,outer\_pri\_hole} = m_{a,inner\_pri\_hole} = 6\% \times m_{a,3} \tag{139}$$

(5) The ratio of flow rate of hole to annular passage:

$$\alpha_{outer\_liner} \tag{140}$$

$$\alpha_{inner\_liner} \tag{141}$$

(6) The inner and outer primary hole jet angle:

$$\theta_{outer\_pri\_hole} = \theta_{inner\_pri\_hole} = \arcsin \sqrt[2]{\frac{C_{d,outer}}{C_{d,\infty}}} \tag{142}$$

(7) The effective areas of primary holes of outer primary holes and inner primary holes:

$$A_{outer\_pri\_hole,eff} = A_{inner\_pri\_hole,eff} = \frac{m_{outer\_pri\_hole}}{\rho_3 \times V_j} \tag{143}$$

(8) The physical areas of primary holes of outer primary holes and inner primary holes:

$$A_{outer\_pri\_hole,eff} = A_{inner\_pri\_hole,eff} = \frac{A_{outer\_pri\_hole,eff}}{C_{d,outer\_pri\_hole}} \tag{144}$$

(9) The number of outer primary holes is assumed to be equal to two times of the number of fuel injectors, namely 36. The same as the inner primary holes.

$$N_{outer\_pri\_hole} = N_{inner\_pri\_hole} \tag{145}$$

(10) The physical diameters of primary holes of outer primary holes and inner primary holes:

$$d_{outer\_pri\_hole} = d_{inner\_pri\_hole} = \sqrt[2]{\frac{4 \times A_{outer,pri\_hole,phy}}{\pi \times N_{outer,pri\_hole}}} \tag{146}$$

Dilution Hole Design

iv. Functions and Requirements

The dilution zone is use to achieve a desired temperature distribution required by turbine components. The quality of temperature distribution is generally measured by “pattern factor (PF)”.

For nozzle guide vanes, they withstand the maximum temperature at the combustor outlet plane. This maximum temperature effects directly on the life of the guide vanes. A pattern factor is expressed:

$$PF = \frac{T_{max} - T_{t4}}{T_{t4} - T_{t3}} \tag{147}$$

For rotor blades, they withstand the correspondingly circumferential average temperature at each radius due to rotation. Generally, the arithmetic mean method is used to calculate the circumferential temperature. Another expression is defined to describe the quality of temperature distribution.

$$PF = \frac{T_{mr} - T_{t4}}{T_{t4} - T_{t3}} \tag{148}$$

In general, 20~40% percentage of total combustor inlet air is used for dilution. The dilution air available for modern high temperature rise combustor will be reduced further because more air is used for combustion and cooling. The reverse flow combustor design has more difficulties because it has large ratio of surface to volume and part of air is need to cool the inner and outer turn sections.

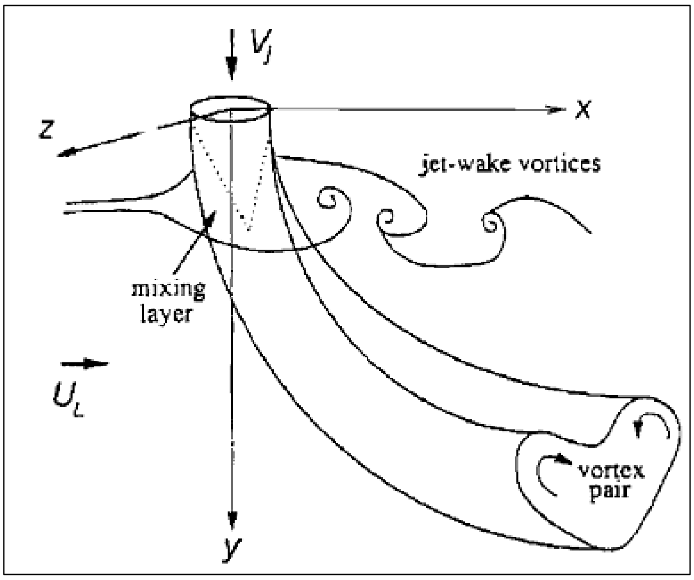


Figure 5 Jet from a hole into a cross flow

The dilution air is generally admitted into the liner to mix with the combustion products through dilution holes. Figure 5 shows the typical jet discharge from a reservoir to a cross stream. The injection blocks the mainstream flow. This blockage effect will produce a pressure differential which results in deformation of the jet. The jet penetration of dilution air has significant effect on pattern factors at the combustor outlet plane. The penetration should be 0.4 liner dome height [Mohammad, 2009; Lefebvre, 1989; Gupta et al., 1984]. The maximum penetration can be calculated from the flowing expressions [Lefebvre, 1989; Gupta et al., 1984]:

$$Y_{\max} = 1.15 \times d_j \times J^2 \times \sin\theta \tag{149}$$

$$Y_{\max} = 1.25 \times d_j \times J^2 \times \frac{m_g}{m_g + m_j} \tag{150}$$

v. Design Procedure

(1) The pressure loss along the liner is assumed to equal to the overall pressure loss

$$\Delta P_{liner} = P_{t3} - P_{t4} \tag{151}$$

(2) The jet velocity:

$$V_j = \sqrt[2]{2 \times \frac{\Delta P_{liner}}{\rho_3}} \tag{152}$$

(3) The hole pressure loss factor:

In outer annular passage, the hole pressure loss factors are calculated:

$$K_{outer\_liner} = \frac{\frac{1}{2} \times \rho_j \times V_j^2}{\frac{1}{2} \times \rho_j \times V_{pass}^2} \tag{153}$$

$$K_{inner\_liner} = \frac{\frac{1}{2} \times \rho_j \times V_j^2}{\frac{1}{2} \times \rho_j \times V_{pass}^2} \tag{154}$$

(4) The air flow rates through outer liner dilution holes and inner liner primary holes are assumed to be 10% of total mass flow rate, respectively.

$$m_{a,outer\_dil\_hole} = m_{a,inner\_dil\_hole} = 10\% \times m_{a,3} \tag{155}$$

(5) The ratio of flow rate of hole to annular passage:

$$\alpha_{outer\_liner} = \frac{m_{a,outer\_dil\_hole}}{3.96} \tag{156}$$

$$\alpha_{inner\_liner} = \frac{m_{a,outer\_dil\_hole}}{0.585} \tag{157}$$

(6) The inner and outer dilution jet angles:

$$\theta_{outer\_dil\_hole} = \theta_{inner\_dil\_hole} = \arcsin \sqrt[2]{\frac{C_{d,outer\_dil\_hole}}{C_{d,\infty}}} \tag{158}$$

(7) The effective areas of outer dilution holes and inner dilution holes:

$$A_{outer\_dil\_hole,eff} = A_{inner\_dil\_hole,eff} = \frac{m_{outer\_dil\_hole}}{\rho_3 \times V_j} \tag{159}$$

(8) The physical areas of outer dilution holes and inner dilution holes:

$$A_{outer\_dil\_hole,phy} = A_{inner\_dil\_hole,phy} = \frac{m_{outer\_pri\_hole}}{C_{d,outer\_dil}} \tag{160}$$

(9) The number and the diameter of dilution holes:

The Cranfield method is used to predict the number and the diameter of dilution holes [9-10]. The method is based on the following equation:

$$n_d \times d_j^2 = 15.25 \times m_j \times (\frac{P \times \Delta P_{liner}}{T_3})^{-0.5} \tag{161}$$

Hence,

(11) The physical diameters of outer dilution holes and inner dilution holes:

$$d_{outer\_dil\_hole} = d_{inner\_dil\_hole} = \frac{d_j}{\sqrt[2]{C_d}} \tag{162}$$

(12) The penetration of dilution holes.

The following equation is recommended to predict the maximum penetration of dilution holes:

$$Y_{max} = 1.25 \times d_j \times J^{0.5} \times m_g / (m_g + m_j) \tag{163}$$

Hence,

$$Y_{max} = 1.25 \times d_j \times J^{0.5} \times m_g / (m_g + m_j) \tag{164}$$

A.3.6 Performance evaluation

After combustor is designed, the performance has been evaluated by empirical equations proposed in the literature. The main parameters include the Sauter Mean Diameter (SMD), combustion efficiency, combustor residence time and emissions.

SMD

The diameter of droplets has significant effect on the combustion efficiency, pattern factor, and the pollutant emissions, etc. A prefilming air blast atomizer is designed to provide the fuel to the combustor. This air blast atomizer is similar to the configuration which has been studied by El Shanawany and Lefebvre [1980].

According to the experimental and analytical results, the authors gave the following equation to predict the SMD for prefilming air blast atomizers:

$$\frac{SMD}{D_h} = (1+\frac{m_f}{m_{a,ato}})\times[0.33\times(\frac{\sigma_f}{\rho_a \times V_a^2 \times D_p})^{0.6}\times(\frac{\rho_f}{\rho_a})^{0.1} + 0.068\times(\frac{\mu_f^2}{\sigma\times\rho_f \times D_p})^{0.5}] \quad (165)$$

The SMD is proportional to 0.43 power of the hydraulic diameter. This correlation is suitable to the low viscosity liquid.

Combustion Efficiency

The combustion efficiency, η, is defined as the ratio of heat released in combustion to the heat supplied. Based on burning velocity model, Lefebvre combined the effects of combustor operating pressure and temperature, and combustor characteristic dimensions to evaluate combustion efficiency. This is the famous “θ” parameter [Lefebvre, 1989; Gupta et al., 1984].

$$\eta = f(\theta) = f(\frac{P_3 \times A_{ref} \times D_{ref}^{0.75} \times \exp(\frac{T_3}{300})}{m_3}) \quad (166)$$

$$\theta = \frac{P_3^{1.75} \times A_{ref} \times D_{ref}^{0.75} \times \exp(\frac{T_3}{300})}{m_3} \quad (167)$$

Pattern Factor

The quality of combustor outlet temperature distribution can be measured by the pattern factor which is essential to the life of turbine components. Two factors have significant effects on the pattern factor. One is the total liner length; another is the pressure loss across the liner. The former is related to the length and time used for dilution; the latter correlates to jet penetration of dilution air.

Based on a large amount experiment results, Lefebvre correlated the pattern factor as:

$$\frac{T_{max}-T_4}{T_3-T_4}=f\left(\frac{L_{liner}}{H_{dome}}\times\frac{\Delta P_{liner}}{q_{ref}}\right) \tag{168}$$

For annular combustors, the following equation can be used to predict the pattern factor [Lefebvre, 1989; Gupta et al., 1984].

$$\frac{T_{max}-T_4}{T_3-T_4}=1-\exp\left(-0.05\times\frac{L_{liner}}{H_{dome}}\times\frac{\Delta P_{liner}}{q_{ref}}\right)^{-1} \tag{169}$$

Lefebvre [Lefebvre and Ballal, 2010] proposed that the pattern factor reduces with the increase of $\frac{L_{liner}}{H_{dome}}\times\frac{\Delta P_{liner}}{q_{ref}}$, but the variation of pattern factor is not evident when $\frac{L_{liner}}{H_{dome}}\times\frac{\Delta P_{liner}}{q_{ref}}$ is over 70.

A.3.7 Emissions

NO_X

Based on analyses of experiment results on different conventional aero engine combustors, Lefebvre derived the following correlation to predict the NOx [Lefebvre, 1981].

$$EI_{NOx}=\frac{9\times10^{-8}\times\left(\frac{P_{3t}}{1000}\right)^{1.25}\times V_c\times\exp(0.01\times T_{st})}{m_{a,3}\times T_{pz}} \tag{170}$$

CO

Lefebvre also derived a correlation to predict the CO formation [Lefebvre, 1981]:

$$EI_{CO}=\frac{86\times m_{a,3}\times T_{pz}\times\exp(-0.00345T_{pz})}{(V_c-V_e)\times(\Delta P/P_3)^{0.5}\times(P_3/1000)^{1.5}} \tag{171}$$

The term V_e means the volume occupied for fuel evaporation. It is obtained as:

$$V_e=\frac{0.55\times m_{pz}\times SMD^2}{\rho_{g,pz}\times\lambda_{eff}} \tag{172}$$

UHC

Rizk and Mongia presented the following correlation to predict UHC formation [Rizk and Mongia, 1994]:

$$EI_{UHC} = \frac{0.755 \times 10^{11} \times \exp(9756 / T_{pz})}{P_3^{2.3} \times (\tau - \tau_{evp})^{0.1} \times (\Delta P / P)^{0.6} \times (\Delta P / P_3)^{0.6}} \tag{173}$$

The term is defined as:

$$\tau_{vep} = \frac{SMD^2}{\lambda_{eff}} \tag{174}$$

A.3.8 Summary of Reverse Flow Combustor Design

Air blast atomizers are designed to achieve low emissions. The ratio of air to fuel by mass is sufficient to ensure good atomization. Based on some research results available, the rotational directions of fuel and air are determined.

In order to shorten the length of the combustor, advanced swirl cup is used. This swirl cup consists of a primary swirler, a radial secondary swirler, a venturi and a flare. The application of swirl cup is also expected to achieve high combustion efficiency and good ignition performance, etc. Different cooling schemes are used to cool the combustor. The impingement cooling scheme is used for dome cooling, film cooling scheme for liner cooling and effusion cooling scheme for turn section cooling.

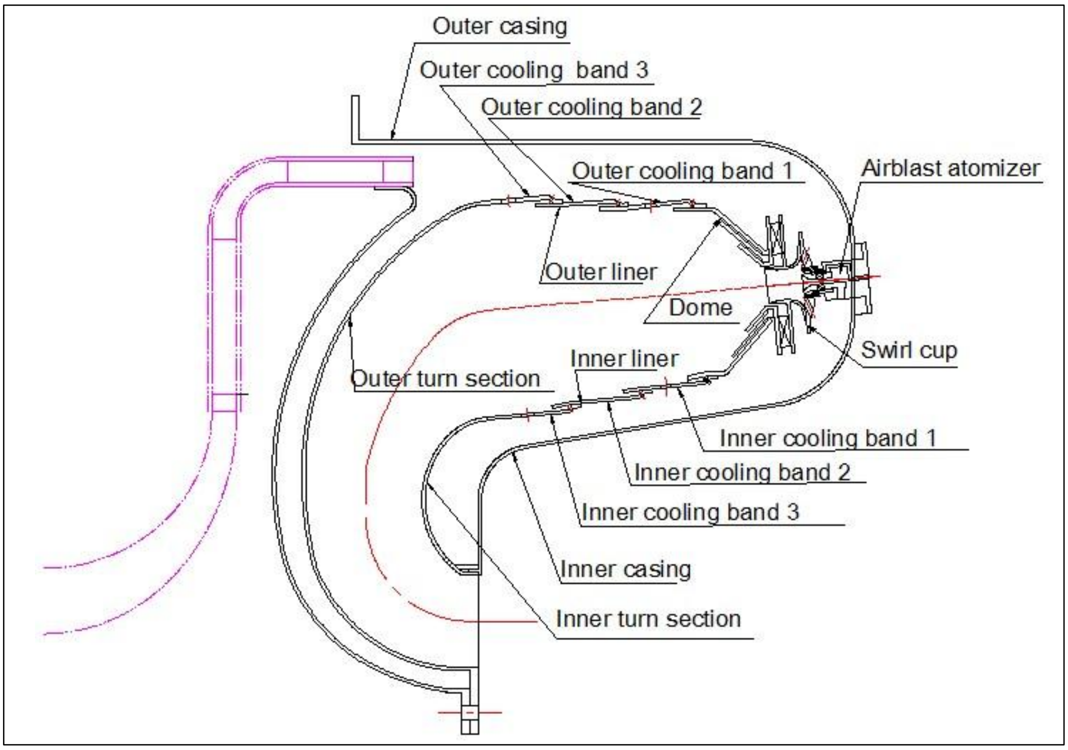


Figure A-25 Schematic diagram of the developed combustor [Khandelwal, 2011(a)]

The design of primary holes and dilution holes is also presented, including the type of the holes, the number of the holes and the hole space ratio, etc. For dilution zone design, two methods have been discussed. One is the Cranfield design method [Mohammad, 2009]; the other is the NASA method [Mohammad, 2009].

A.3.9 Conclusions

The presented paper gives a preliminary design methodology of a reverse flow gas turbine combustor for helicopter engines. To the author's knowledge no available literature explains the procedure for designing a reverse flow combustor. This paper presents combustor as a detailed package including the fuel injector, swirler, cooling system, primary holes and dilution holes. The liner is designed in parts with the cooling system and air ingestion holes. The standard application of reverse flow combustors is along with the centrifugal compressor units. Hence, diffuser design is not considered since the centrifugal compressors inherently have a very low exit Mach number. The selections of components are done on the basis of simplicity, applicability and emission considerations. Design adaptation of three different cooling schemes are discussed, which are, impingement cooling scheme for the dome, film cooling scheme for liner and effusion cooling scheme for turn sections. A design summery is presented for an overall understanding. The overall combustor performance is evaluated by some empirical correlations. Attempts are also made to predict the emission of reverse flow combustor in this paper.

Appendix B Novel Hybrid Diffuser

ABSTRACT

Significant development in the gas turbine technology has brought about an increase in the performance requirements for modern engines. This has generated a significant interest in researching into implementation of novel technologies for various engine components, which will allow for the design of engines to match the new performance requirements. One such technology is the use of hybrid diffusers in gas turbine combustors against conventional combustors like dump diffusers. The hybrid diffuser concept has been around for a while and has the potential of giving a greater performance than conventional diffusers. However, due to limited information available in the public domain, not much has been fully understood about the mechanism of the hybrid diffuser concept. Much of the previous work done on hybrid diffusers are done on designs having a vortex chamber bleed, based on the belief that vortex chambers helps to stabilize the flow separation. However, this paper takes looks into the proposition that the primary mechanism of a hybrid diffuser is the air bleed rather than the vortex chamber itself. This paper looks at a comparative study between a hybrid diffuser with a vortex chamber and that of a new hybrid diffuser design where the vortex chamber is replaced with a duct bleed. The Diffuser Pressure Loss, Bleed Pressure Loss and Pressure Recovery Coefficients of each were analyzed at different bleed rates. The results yield that the duct bleed hybrid diffuser has a similar performance to that of a vortex chamber hybrid diffuser. However, it was observed that a duct bleed needed even less bleed air to achieve a good performance and thus suggesting Walker's proposition to be true that a vortex chamber is not a necessary configuration in hybrid diffuser.

Also included is the analysis of performance characteristics of various configurations of duct bleed hybrid diffusers. These include changes in post-diffuser divergence angles, bleed angles and changes in the dump gap. Changes in Diffuser angles do give a better performance but at the expense of

larger amount of air bleed from the mainstream. Reducing the bleed angle made no significant gain to the diffuser performance but improved the quality of bleed air which can be used for other purposed like turbine cooling. The paper also discusses as to why there hasn't been any real practical application of this novel technology until now and also suggests specific area where hybrid diffuser could provide immediate benefits.

INTRODUCTION

With the recent rapid development in gas turbine technology and the desire to have a higher engine performance, a substantial interest has been generated in applying evolutionary technology to each of the components in the engine. In modern axial flow compressors, the objective to achieve a high pressure rise with fewer compressor stage results in a relatively higher flow velocity. This flow velocity can reach a value of around 170 m/s or more [Lefebvre and Ballal, 2010]. The high performance requirement of the turbines and compressors of a modern engine, results in a higher diffuser inlet Mach number. The combustion process with conventional diffusers, a faired or a dump diffuser, can be inefficient at such high speeds and a significant pressure loss can occur across the flow of the working fluid in the diffuser, upstream of the combustor chamber.

Faired Diffuser, as shown in Figure B-1 (a), was mostly used in the early years of gas turbine engines. Though it provided a great advantage in terms of a relatively low pressure loss across the flow, but the design couldn't accommodate any stall or flow separation of the flow [Lefebvre and Ballal, 2010]. In order to avoid stall and the condition of flow separation, faired diffusers designs usually had small divergence angles and longer lengths. Additionally, the flow in the faired diffusers was found to be very sensitive to the inlet velocity profile and also the diffuser itself was subject to recurrent manufacturing and thermal distortion problems. Due to these limitations faired diffuser were replaced by dump diffusers, which are now widely used in the industry.

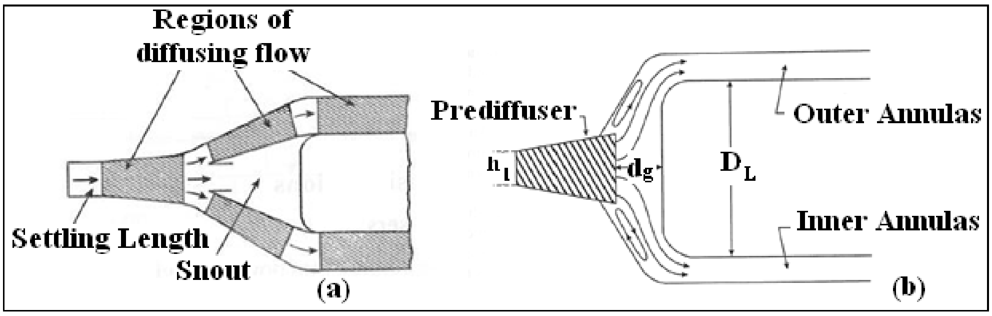


Figure B-1 (a) Faired Diffuser (b) Dump Diffuser [Lefebvre, 2010].

In a dump diffuser, as shown in Figure B-1 (b), the air-flow at the inlet diffuses through a short conventional diffuser called pre-diffuser. At the exit of this pre-diffuser, the velocity of the flow can be reduced to about 60% of the inlet flow velocity. The air is then "dumped" and the flow is divided into two streams surrounding the liner. The vortexes shown in Figure B-1(b) assist to maintain a stable and uniform flow in the dumped region. Due to the flow being dumped and passing through a larger turning angle, a higher pressure loss will occur when compared to faired diffusers. But on the other hand the disadvantage in pressure loss, due to higher divergence angle, is offset by a shorter length of the dump diffuser. Also the dump diffusers tend to provide continuous stable flow patterns, which are insensitive to the inlet velocity profile and manufacturing and thermal tolerances.

However, as the diffuser inlet velocity for modern engines will tend be even higher, it may not be feasible to use either of the two conventional diffusers. When selecting a diffuser, the choice largely depends on its application. From the designer's point of view, an ideal diffuser is one that achieves the required velocity reduction in the shortest possible length, with minimum loss in total pressure and with uniform and stable flow conditions at its outlet [Lefebvre and Ballal, 2010]. If either of the two conventional diffusers were to be used for modern engines, it may not be feasible as there will be a significant in pressure loss across the flow to achieve the desired velocity reduction of a much faster airflow. To minimize the pressure loss with conventional designs, it will either require the diffuser to be longer in length or to have a higher divergence angle. Along with the associated pressure losses, incorporating the necessary adjustments to the designs will cause a reduction in thrust to weight ratio and may increase the costs in operating the modern gas turbine engine.

One novel diffuser concept, developed by Adkins et al. [1981] and Adkins and Yost [1983], shows great potential to be used as a substitute for application in modern gas turbine combustors as it provides an improvement in static pressure recovery along with a notable reduction in diffuser lengths. This diffuser is called a hybrid diffuser and is derived from the concept of a vortex controlled diffuser. A vortex-controlled diffuser is one where the working fluid experiences a sudden expansion across an obstruction. Figure B-2 illustrates a tubular configuration of a vortex controlled diffuser. The working fluid, air in this case, flows across an aperture and instantaneously faces a sudden expansion across the fence. A small amount of the mainstream air is bled out through the aperture. The mechanism of such a diffuser is not very well understood but a study done by Adkins [1975] suggests that the suction effect of the bleed air helps to prevent the flow from stalling after the sudden expansion.

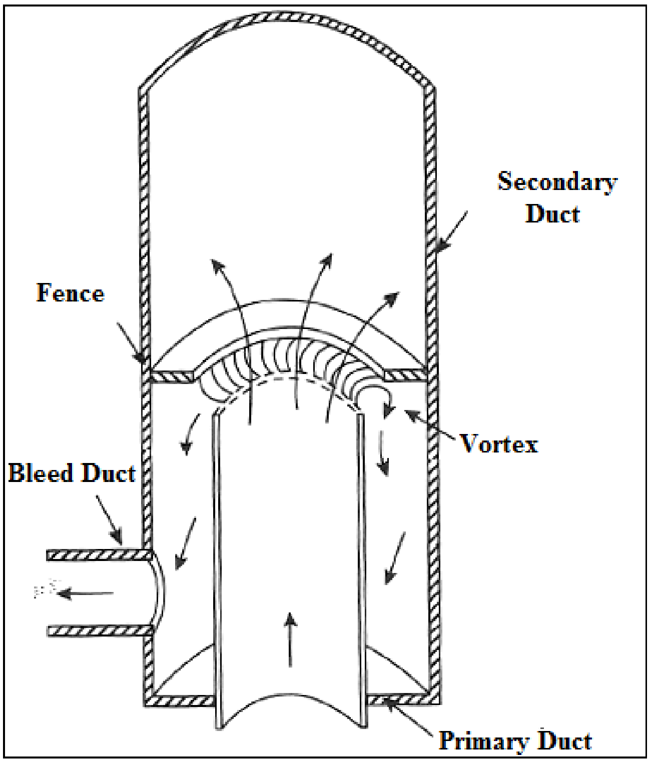


Figure B-2 Vortex-controlled diffuser of tubular configuration [Adkins, 1975].

Adkins et al. [1981] provides a useful explanation about the flow mechanism of a vortex-controlled diffuser which is shown in Figure B-3. He suggests, the flow through the diffuser is divided into two streams, “a” and “b”. The flow from stream “a” will be drawn into the vortex chamber with considerable acceleration (external suction applied), while stream “b” will flow down the diffuser to a region of greater static pressure and therefore decelerate. The velocity differential between these two streams produces a shearing action and creates an extremely turbulent layer. This phenomenon results in an energy transfer from the bleed stream “a” to the mainstream “b”. Therefore the stream “b” can flow down through the diffuser without stall. Also the main stream flow is further aided by the formation of “Coanda bubble” behind the fence, just as the cusp vortex. There is a minimum bleed requirement for the vortex-controlled diffusers [Adkins et al., 1981].

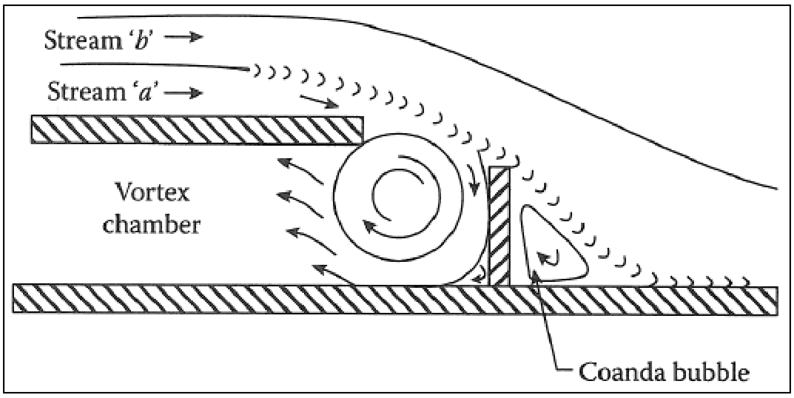


Figure B-3 Flow Mechanism through a Vortex Chamber [Adkins et al., 1981]

One of the main drawbacks of a vortex-controlled diffuser is that it requires a minimum amount of bleed air [Adkins et al., 1981] to achieve the desired performance. When the percentage of bleed is low the pressure recovery is low. As we start to increase the bleed rate, there will be a slight increase in the value of pressure recovery. Then as we increase the percentage of bleed to the minimum bleed requirement, a sudden increase in pressure recovery will be observed [Adkins et al., 1981]. For modern gas turbine engines a considerable amount of bleed air will be required to achieve the enhanced performance. To overcome this problem of high bleed rate requirement Adkins and Yost, 1979; proposed the concept of a “hybrid diffuser”, which is a combination of vortex-controlled diffuser and wide angle conventional diffuser. A hybrid diffuser positions a conventional diffuser immediately downstream of the vortex bleed diffuser, as shown in Figure B-4.This concept was first given by Sutherland, 1972.

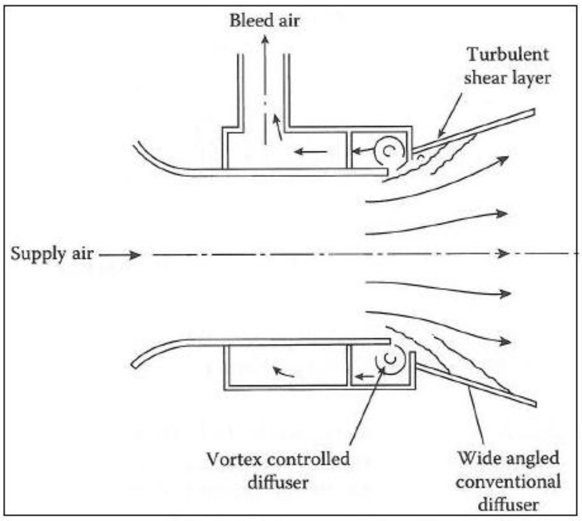


Figure B-4 Hybrid Diffuser [Adkins et al., 1981]

Experimental tests indicate hybrid diffusers to have a superior performance than the conventional ones, even without bleed [Adkins et al., 1981]. Therefore, the high performance hybrid diffuser has the potential to be a good prospect for use in modern gas turbine engines. But one problem that may arise in the implementation of a hybrid diffuser is of bleed air. As one of the possible uses of bleed is to use it for the cooling of the turbines, but the pressure of the bleed air is too low to do so. A solution to this problem was proposed by a new design (Figure B-5) by Adkins and Yost [1983] where a pre-diffuser with a small area ration is attached at the upstream of the bleed vortex controlled diffuser. In this new configuration as the air reaches the location of the bleed, the pre-diffuser installation allows the air to have a slightly higher pressure. Therefore, allowing the bleed air to be considered as a viable choice for use in turbine cooling. The main diffusion process is then carried out in the downstream section to achieve the required performance, i.e. the desired velocity reduction. One possible shortcoming of such a design is the extra length required by the pre-diffuser. Nevertheless, it shows a greater promise for use in actual gas turbine engines.

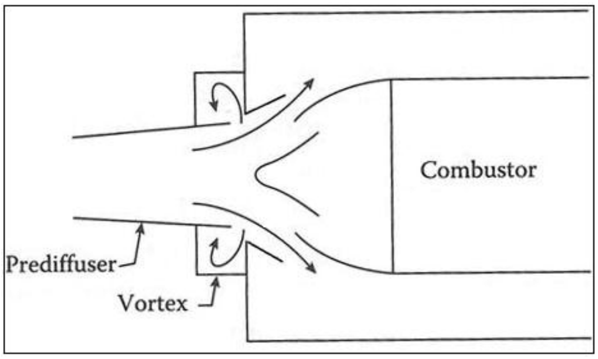


Figure B-5 Hybrid Diffuser, with a pre-diffuser [Adkins, 1975]

Most of the previous studies on hybrid diffusers have been on designs having a vortex chamber bleed. All the hybrid diffusers studied by Adkins et al. [1981], Adkins and Yost [1983], Adkins [1975] and Adkins and Yost [1979] were based on the assumption that the vortex chambers help to stabilize the flow separation and are crucial to the design of hybrid diffusers. However, Walker [2002] argued that the primary mechanism of hybrid diffuser was the bleed of the flow rather than the use of vortex chamber. Therefore this study undertakes a comparative study between a hybrid diffuser with a vortex chamber bleed as against without a vortex chamber bleed. In the latter configuration the vortex chamber bleed is substituted for with a duct bleed, having a bleed angle of 90 degrees. A pre-diffuser has been incorporated in both the hybrid diffusers considered in this study. Various changes were then made to the duct bleed hybrid diffuser configuration to analyze it effects on the diffuser performance characteristics. These included effects of change in duct bleed angle, divergence angle and the dump gap. The comparative study between these different diffusers has been performed using computational fluid dynamics. The diffuser parameters analyzed for each design, are the Pressure Recovery Coefficient and Pressure Loss Coefficient.

Methodology

Hybrid Diffuser Design

The hybrid diffuser design used in this study was based on experiments carried out by Adkins et al. [1981], Adkins and Yost [1981]. In the experiments done by Adkins and Yost [1983] the diffuser inlet Mach number was 0.35. Therefore, the design of vortex controlled steps on both inner and outer walls of the diffuser was based on the available experimental data [Adkins et al., 1981]. Figure B-6 shows a schematic diagram of the final hybrid diffuser design. The corresponding values of pre-diffuser half divergence angle and post-diffuser half divergence angle are 6.1 degrees and 9 degrees accordingly. The area ratio of the Hybrid Diffuser is 2.5.

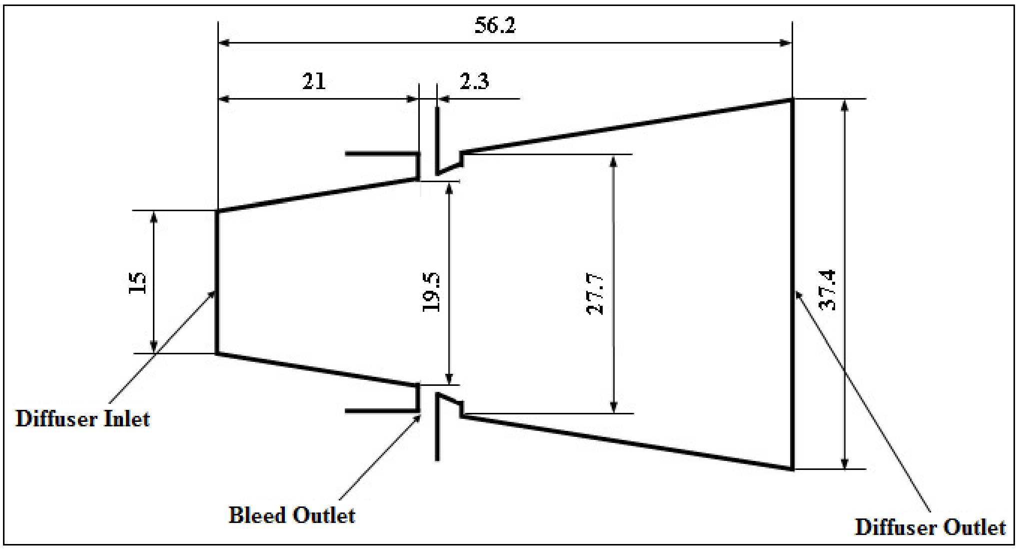


Figure B-6 Hybrid Diffuser Dimensions [Khandelwal, 2011(b)]

Different Cases Considered in this Study

Five different cases of hybrid diffusers have been considered in this study. Figure B-7 shows all the different configurations considered to study these cases. All the design parameter of the four cases is same unless mentioned.

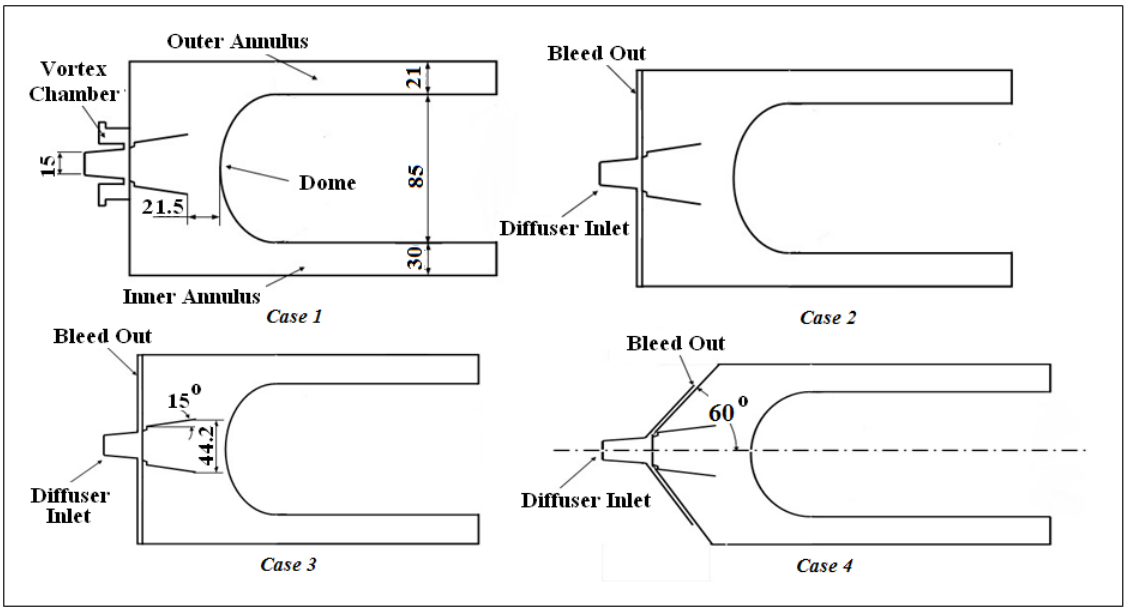


Figure B-7 Different Hybrid Diffuser Configurations [Khandelwal, 2011(b)]

In Case 1, a vortex chamber bleed was incorporated in the hybrid diffuser design. In Case 2, the vortex chamber was replaced with a narrow duct, perpendicular to the stream, to extract the bleed air. This allowed for the comparison of performance characteristics of hybrid diffusers with and without a vortex chamber bleed. In Case 3, the post-diffuser divergence angle of the duct bleed hybrid diffuser is varied to study the influence of changes in divergence angle on the hybrid diffuser performance. For Case 3, the post diffuser divergence angle is changed to 15 degrees, from the 9 degrees used in Case 2. The length to width ratio was kept constant and the new area ratio for this case changed to a value of 2.9. In Case 4, the duct bleed angle was varied to analyze the influence it had on performance of the hybrid diffuser and the pressure characteristic of the bleed air. The bleed angle for Case 4 was changed to 60 degrees. In Case 5, the influence of dump gap on the performance of the duct bleed hybrid diffuser was investigated. The configuration for Case 5 is the same as that of Case 2 shown in Figure B-7. The dump gap was changed from 21.55 mm to 25 mm and 30 mm, respectively.

Other Design Parameters

The objective of the study was to primarily research into how hybrid diffuser configurations without a vortex chamber bleed influenced the diffuser characteristic. Therefore no struts were included in the upstream and the air admission into the liners was not accounted for in the design. The dome used in all of the four cases was taken as a simple round headed shape. The inner and outer annulus dimensions have been kept the same. The length of the dump gap was derived using Bradshaw’s study [Bradshaw, 2002] to minimize the pressure loss of the flow leaving the diffuser, the dump gat ratio should be around a value of 1.43. This value of 1.43 was used in the calculation of dump gap and dump gap value calculated in this design is 21.55mm

CFD Simulation and Validation

Numerical simulations were carried out using commercial CFD code ANSYS FLUENT 12.1 [2010]. A diffuser model from Adkins experiment [Adkins et al., 1981] was selected to validate the choice of different models from CFD. The geometry simulated is illustrated in Figure B-8. The overall area ratio is 2.5, and the vortex controlled area ratio is 1.2. The post diffuser divergence angle in this model is 20 degrees.

To avoid simulation inaccuracy caused by the sudden diffusion from the inlet, a 5mm setting length was located upstream of the diffuser inlet. Simulation results of the validation model showed that the result derived from SST k-ω model had the closest pressure recovery coefficient characteristic to that of the experimental data. Furthermore, as the SST k-ω model is also more suitable for adverse pressure gradient conditions, especially with separation [Menter, 1994], it was selected to simulate the various proposed cases of the hybrid diffuser.

The operating pressure was set to 5068192 Pa, same as the diffuser inlet total pressure. The value for total temperature was set at 998 K. The working medium was selected to be ideal gas and the specific heat capacity was set to 1140 J/ (kg*K).

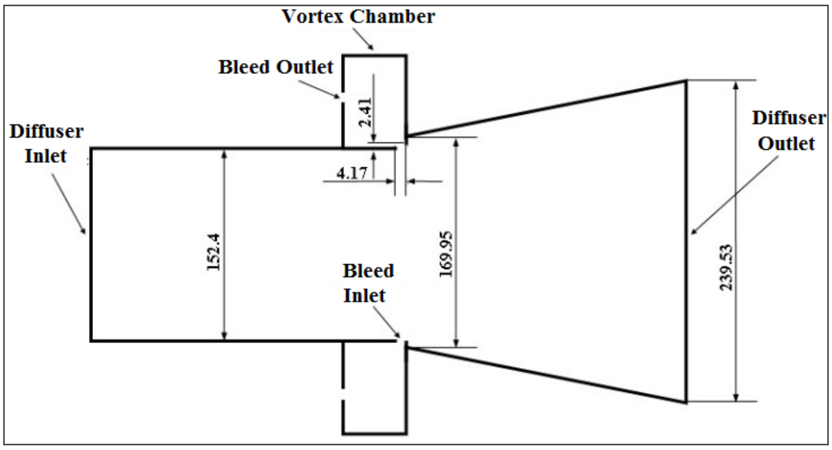


Figure B-8 Validation Model from Adkins' experiment [Adkins et al., 1981]

Simulation Cases

The various hybrid diffuser designs shown in Figure B-7 are simulated to investigate the influence of bleed air on each of the designs. The bleed air percentage of the main flow is varied in 1% increments from 1% to 5 %. All the studied cases are tabulated in Table 1.

Table 1 – Simulated Cases and Parameters

Case	AR	Post Diffuser Half Divergence Angle	Bleed Rate	Dump Gap (mm)	Bleed Angle	Note
Case 1-1	2.5	9 º	1%	21.5	-	Hybrid Diffuser with Vortex chamber and fence
Case 1-2	2.5	9 º	2%	21.5	-	
Case 1-3	2.5	9 º	3%	21.5	-	
Case 1-4	2.5	9 º	4%	21.5	-	
Case 1-5	2.5	9 º	5%	21.5	-	
Case 2-1	2.5	9 º	1%	21.5	90 º	Duct Bleed Hybrid Diffuser (Without vortex chamber, but with fence)
Case 2-2	2.5	9 º	2%	21.5	90 º	
Case 2-3	2.5	9 º	3%	21.5	90 º	
Case 2-4	2.5	9 º	4%	21.5	90 º	
Case 2-5	2.5	9 º	5%	21.5	90 º	
Case 3-1	2.9	15 º	1%	21.5	90 º	Variation in Divergence Angle of Duct Bleed Hybrid Diffuser
Case 3-2	2.9	15 º	2%	21.5	90 º	
Case 3-3	2.9	15 º	3%	21.5	90 º	
Case 3-4	2.9	15 º	4%	21.5	90 º	
Case 3-5	2.9	15 º	5%	21.5	90 º	
Case 4-1	2.5	9 º	1%	21.5	60 º	Variation in Bleed Angle of Duct Bleed Hybrid Diffuser
Case 4-2	2.5	9 º	2%	21.5	60 º	
Case 4-3	2.5	9 º	3%	21.5	60 º	
Case 4-4	2.5	9 º	4%	21.5	60 º	
Case 4-5	2.5	9 º	5%	21.5	60 º	
Case 5-1	2.5	9 º	1%	21.5	90 º	Variation in Dump Gap Duct Bleed Hybrid Diffuser
Case 5-2	2.5	9 º	2%	25	90 º	
Case 5-3	2.5	9 º	3%	30	90 º	

The main parameters analyzed are the Pressure Recovery Coefficient (C_P) and the Pressure Loss Coefficient (λ) [Lefebvre and Ballal, 2010]. The former being a function of the static pressure at both the inlet and outlet of the diffuser, while the latter being a function of the total pressures (Figure B-9). Both the coefficients are computed using the following equations:

$$C_P = \frac{(p_2 - p_1)}{q_1} \tag{175}$$

$$\lambda = \frac{(\bar{P}_1 - \bar{P}_2)}{q_1} \tag{176}$$

$$q = \frac{\rho u^2}{2} \tag{177}$$

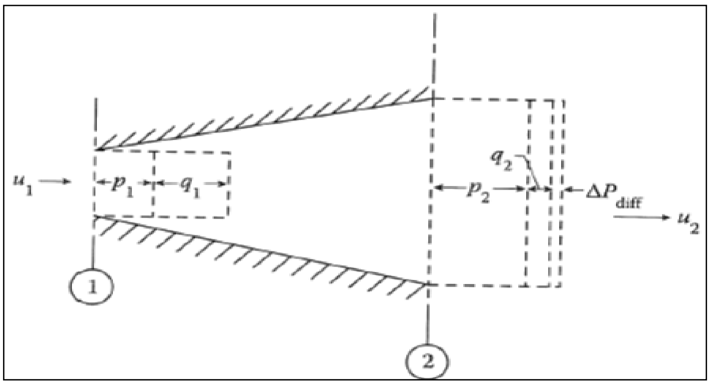


Figure B-9 Energy Conversion in a Diffuser [Lefebvre, 2010]

The higher the Pressure Recovery Coefficient and lower the Pressure Loss Coefficient, the better will be the performance of the diffuser. The total pressure in Eqn. 2 is mass weighted.

Results and discussions

Effect of different parameters on diffuser pressure loss coefficient

It can be observed from Figure B-10 that diffuser pressure loss coefficient for vortex-bleed hybrid diffuser is similar (Case 1) to duct-bleed hybrid diffuser (Case 2). When post-diffuser half divergence angle was enlarged to 15º (Case 3) instead of 9º (Case 2), keeping rest of the parameters same, it is observed that diffuser pressure loss coefficient increases with increase in divergence angle. Diffuser pressure loss coefficient decreases substantially in case 3 as the bleed rate is increased till 3 %. Above 3 % bleed rate there is a slight decrease in diffuser pressure loss coefficient.

Change in bleed angle substantially affects the diffuser pressure loss coefficient. It is observed that as the bleed angle is changed from 90º (Case 2) to 60º (Case 4), for 1 % bleed rate, diffuser pressure loss coefficient decreases from 0.108 to 0.084 for case 2 and case 4 for respectively. Similar effect is observed for other bleed rates also. One of the possible reasons for this change is that it is easier to flow into the bleed passages since the angle is smaller. As for Case 5, it was observed that increasing the dump gap dose not decreases the diffuser pressure loss coefficient.

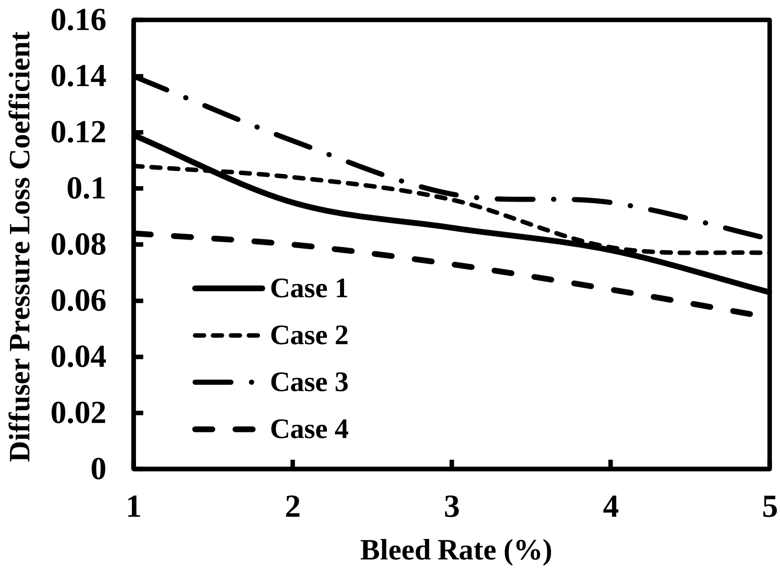


Figure B-10 Variation of Diffuser pressure loss coefficient with bleed rate [Khandelwal, 2011(b)]

Effect of different parameters on bleed pressure loss coefficient

Figure B-11 shows the variation in bleed pressure loss coefficient at different bleed rate for different configuration under this study. Bleed pressure loss coefficient in vortex-bleed hybrid diffuser (Case 1) is substantially high as compared to duct-bleed hybrid diffuser (Case 2). Bleed pressure loss coefficient at 1 % bleed rate is 0.58 and 0.51 for case 1 and case 2 respectively. This shows that diffuser performance would be better if a duct bleed was used instead of a vortex chamber. Change in post-diffuser divergence angle dose not significantly affect bleed pressure loss coefficient. Changing bleed angle substantially affects bleed pressure loss coefficient. When the bleed angle is decreased from 90º (Case 2) to 60º (Case 4), bleed pressure loss coefficient decreases from 0.58 to 0.48 at 1 % bleed rate. As the bleed rate is increased this difference increases substantially. For Case 5, as observed for the diffuser pressure loss coefficient, change in dump gap has no significant affect on the bleed pressure loss coefficient.

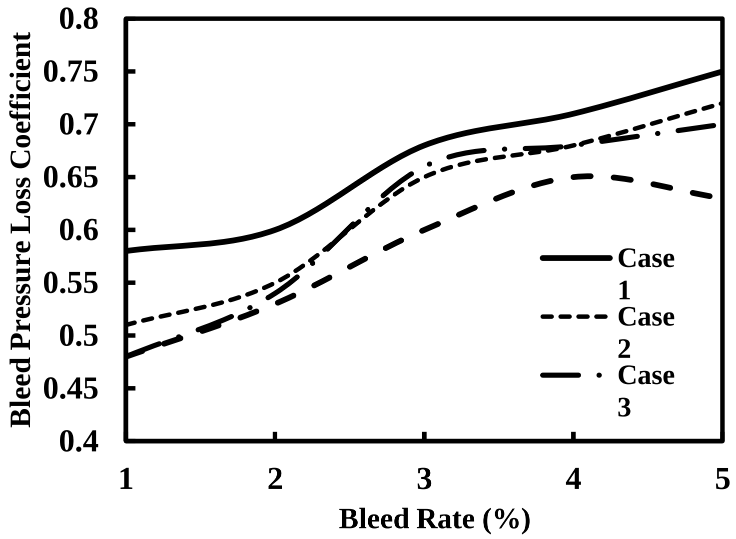


Figure B-11 Variation in Bleed Pressure Loss Coefficient at different bleed rates [Khandelwal, 2011(b)]

Effect of different parameters on Pressure Recovery Coefficient

Results are also compared with experimental results of work done by Adkins et al. [1981]. It is observed that results obtained by Adkins et al. [1981] are substantially similar for 2-3% bleed rate. For case 3 pressure recovery coefficient increases with bleed rate. At bleed rates of 2% and above, case 3 is giving substantially better pressure recovery coefficient then other cases and reference [Adkins et al., 1981] as shown in Figure B-11.

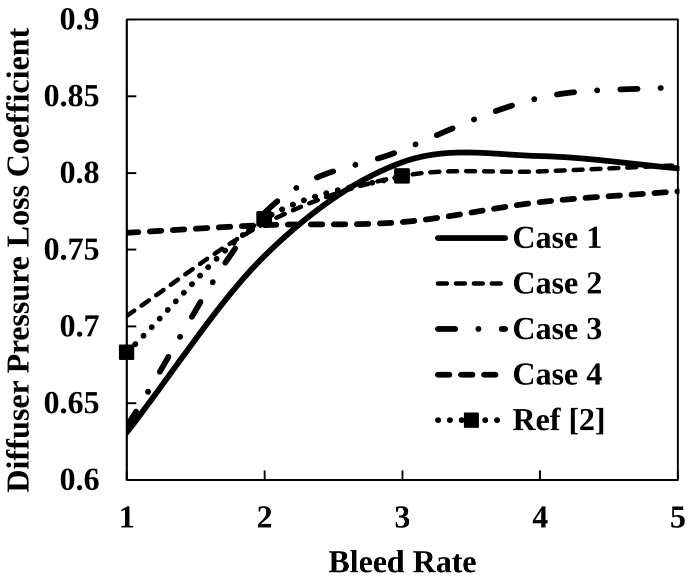


Figure B-12 Variation in Pressure Recovery Coefficient of Diffuser at different bleed rates [Khandelwal, 2011(b)]

From Fig. 16, it can be seen that at bleed rate of 3 %, 4 %, 5 %, the pressure recovery coefficient of vortex-bled (Case 1) and duct-bled (Case 2) hybrid diffuser are quite similar. While in lower bleed rate conditions, the duct-bled diffuser seems to give a better performance than the vortex-bled one. It has been observed that at bleed rate of 2 %, the flow in duct bleed hybrid diffuser tends to be un-separated, whereas, the vortex-bled hybrid diffuser needs 3 % of bled air for getting an un-separated flow. One possible reason for this is that, in the duct-bled configuration, the flow is bled out directly. So the suction effect is stronger than the vortex-bled configuration, and it is relatively easier to drag the main flow to the diffuser wall.

It can be observed from the Figure B-12 that when the post diffuser divergence angle is increased from 9 (Case 2) to 15 (Case 3) the diffuser performance would be better at high bleed rates. As the bleed angle is decreased from 90º (Case 2) to 60º (Case 4), pressure recovery coefficient is higher for case 4 as compared to case 2, till bleed rate of 2 %. Above 2 % bleed rate performance of case 2 is substantially higher than that of case 4. For Case 5, changing the dump gap to lengths of 25 mm and 30 mm, did not affect pressure recovery coefficient substantially.

Summary of results and discussion

The vortex-bled (Case 1) and duct-bled (Case 2) hybrid diffusers have similar performance, and the duct-bled configuration needs less bled air to achieve a good performance. Therefore, the vortex chamber is not a necessary configuration for the hybrid diffuser. The momentum transfer between the bleed air and the main stream is the primary mechanism. Compared to the post-diffuser divergence angle of 9° (Case 2), it is clear that the diffuser performance is better at high bleed rate conditions for post diffuser divergence angle of 15°. This is possibly because that the total area ratio is larger.

It can be seen that at a lower bleed rates percentages, the pressure recovery coefficient is better for case 4, when the bleed angle is 60º, than the previous configurations. But the pressure recovery coefficient increases very slowly with the increase of the bleed rate (Figure B-12). One possible explanation for this is that it is easier for the air to flow into the bleed passages since the angle is smaller. But the suction effect to the main stream is not as strong as it is with configurations having a larger bleed angle. As shown in Figure B-12, one useful benefits of reducing the bleed angle is that, the pressure loss in the bleed passage is relatively smaller than that of larger bleed angles. At 2% and above bleed rates case 3 is giving substantially better pressure recovery coefficient then other cases and reference [Adkins et al., 1981]. Therefore, suggesting that the bleed air is of much better quality and can be useful for turbine cooling or other uses. But the penalty is that it requires more bleed air, which might not be affordable for most modern gas turbine engines. Change in dump gap has no substantial effects on the performance of the diffuser.

Conclusion

In this study, the performance characteristics of hybrid diffuser designs, one with a vortex chamber and the other with the duct bleed. After analyzing the pressure loss and recovery coefficients at various bleed rates it is seen that both the diffusers had similar results. The duct bleed diffuser needed less bleed air to achieve the desired performance. This observation suggests that a vortex chamber is not a necessary feature required for a hybrid diffuser. Therefore as argued by Walker [2002], this study concludes that the primary mechanism that exists in a Hybrid Diffuser is the momentum transfer between the bleed air and the main stream. Increasing the post-diffuser divergence angle of the duct bleed hybrid diffuser does give a greater performance, but this brings along with a penalty with a larger bleed air requirement. The reduction of the bleed angle had no substantial effect on the performance, but gave a better bleed air quality which could be used for purposes like turbine cooling. Change in the Dump gap had no significant performance on the duct bleed hybrid diffuser. Many studies have shown that vortex controlled hybrid diffusers have better performance than conventional diffusers. However till date there is no practical use of hybrid diffusers in gas turbine engines. One possible reason is that nobody wants to take the risk of the performance decrease due to the penalty of the bleeding air. This study has shown that by increasing the bleed angle of duct bleed hybrid diffuser is further investigated, it results in a better quality of bleed air. Therefore this good quality of bleed air can be used for cooling of turbine blades or the trailing edge of nozzle guide vanes.

Appendix C Publications, Patents and Awards

C.1 List of Publication (during PhD)

1. “Hydrogen as a fuel for gas turbine engines with novel micro-mix type combustors” **AIAA Paper Number, AIAA 2011-5806.**

2. “Implication of different fuel injector configurations for hydrogen fuelled micro-mix combustors” in **ASME Turbo Expo 2011, GT2011-46845.**

3. “Flame stabilization studies in a three backward facing step configuration based micro-combustor with premixed methane-air mixtures” **ISABE-2011-1115.**

4. “Design Procedure of a Reverse Flow Combustor for a Helicopter Engine with High Temperature Rise” **SAE Paper Number 2011-01-2562**.

5. “Study of Hybrid Diffusers for Use in Gas Turbine Combustors” **SAE Paper Number 2011-01-2496.**

6. “Study on Nuclear Energy Powered Novel Air Transport Model and its Feasibility study” **AIAA Paper Number, AIAA 2011-6038.**

7. “Design, Evaluation and Performance Analysis of Staged Low Emission Combustors” **ASME Turbo Expo 2012, GT2012-69215.**

8. “Study of novel micro-mix combustors to be used in gas turbines; using hydrogen, hydrogen-kerosene, hydrogen-methane and methane as a fuel” **AIAA Paper Number AIAA-2012-4265.**

9. **“**Preliminary Study of a Novel Gas Turbine Combustor Concept based on Hydrogen Synthesis from Kerosene Reformation” **AIAA Paper Number AIAA-2012-4070.**

10. “A Review of Hydrogen as a Fuel for Future Air Transport” **AIAA Paper Number AIAA-2012-4267.**

11. “Evaluation of Hydrogen Combustors in Gas Turbines” **10th Annual International Energy Conversion Engineering Conference, AIAA Paper Number AIAA-2012-4227.**

12. “Computational study on vortex controlled hybrid diffuser” under internal review for publication with **an International Journal.**

13. “Design and Study on Performance of Axial Swirler for Annular Combustor by changing different design Parameters**” Journal of Power and Propulsion Research** (Submitted for publication)**.**

14. “Preliminary Design and Performance analysis of a Low Emission Aero-Derived Gas Turbine Combustor” **The Aeronautical Journal** (Submitted for publication).

15. “Hydrogen: The future of air transport” **Journal of Progress in Aerospace Sciences** (Submitted for publication) (Preliminary accepted).

C.2 List of Patents

1. A patent on “Design of combustors without dilution zones and with injection of dilution air through NGV”
2. A patent on “Micro-mix combustors for Gaseous Hydrogen”
3. Potential Patent filing on Hydrogen production and its use as an additive in combustors.

C.3 Awards and Achievements

- ASME IGTI Student Award
- RAeS Grant
- IMechE Grant
- Idea of Nuclear Powered Airfleet and Metro concept is widely published in media groups including **Aviation Week and Aerospace America**.

C.4 Abstracts of Publications

C.4.1 Design Procedure of a Reverse Flow Combustor for a Helicopter Engine with High Temperature Rise

(SAE Paper Number 2011-01-2562)

Modern advanced engines are expected to operate with higher combustor temperature rise and lower emissions. These development trends result in more combustor design difficulties. High temperature rise requires more air for complete combustion, hence reducing the amount of cooling air. Emissions consist of CO_2, UHC, NO_x, smoke and water vapor. CO_2 is an unavoidable emission of combustion reaction, whereas emissions of NO_x mainly depend on temperature. A lean primary zone design is required to achieve low NOx emission. Reverse flow combustors have more difficulties because of the presence of turn section which does not exist in other combustor configurations. There are many studies in public domain which talk about design of combustors. But none of them gives detailed guide on designing reverse flow combustor. The objective of this paper is to provide a reverse flow combustor design procedure, with emissions and performance analysis. The combustor designed in this study is expected to be used in advanced helicopter engine. Substantial amount of literature is available on conventional combustor designs which mainly include empirical and semi-empirical models plus experiment test methods. All these combustor design methods focus on the direct flow combustor. In this study, a reverse flow combustor design methodology is proposed. The design procedure includes the combustor sizing, fuel injector design, swirl cup design, air distribution along the liner, primary hole design, dilution zone design and the cooling system design. Final dimensions are also shown in a figure, which have been validated with one of the present combustor design. After finalizing the design of the combustor, overall performance has been evaluated using empirical correlations and equations.

C.4.2 Study of Hybrid Diffusers for Use in Gas Turbine Combustors

(SAE Paper Number 2011-01-2496)

Significant development in the gas turbine technology has brought about an increase in the performance requirements for modern engines. This has generated a significant interest in researching into implementation of novel technologies for various engine components, which will allow for the design of engines to match the new performance requirements. One such technology is the use of hybrid diffusers in gas turbine combustors against conventional combustors like dump diffusers. The hybrid diffuser concept has been around for a while and has the potential of giving a greater performance than conventional diffusers. However, due to limited information available in the public domain, not much has been fully understood about the mechanism of the hybrid diffuser concept. Much of the previous work done on hybrid diffusers are done on designs having a vortex chamber bleed, based on the belief that vortex chambers helps to stabilize the flow separation. However, this paper takes looks into the proposition that the primary mechanism of a hybrid diffuser is the air bleed rather than the vortex chamber itself. This paper looks at a comparative study between a hybrid diffuser with a vortex chamber and that of a new hybrid diffuser design where the vortex chamber is replaced with a duct bleed. The Diffuser Pressure Loss, Bleed Pressure Loss and Pressure Recovery Coefficients of each were analyzed at different bleed rates. The results yield that the duct bleed hybrid diffuser has a similar performance to that of a vortex chamber hybrid diffuser. However, it was observed that a duct bleed needed even less bleed air to achieve a good performance and thus suggesting Walker's proposition to be true that a vortex chamber is not a necessary configuration in hybrid diffuser. Also included is the analysis of performance characteristics of various configurations of duct bleed hybrid diffusers. These include changes in post-diffuser divergence angles, bleed angles and changes in the dump gap. Changes in Diffuser angles do give a better performance but at the expense of larger amount of air bleed from the mainstream. Reducing the bleed angle made no significant gain to the diffuser performance but improved the quality of bleed air which can be used for other purposed like turbine cooling. The paper also discusses as to why there hasn't been any real practical application of this novel technology until now and also suggests specific area where hybrid diffuser could provide immediate benefits.

C.4.3 Design, Evaluation and Performance Analysis of Staged Low Emission Combustors

(ASME Paper No. GT2012-69215)

The most uncertain and challenging part in the design of a gas turbine has long been the combustion chamber. There has been large number of experimentations in industries and universities alike to better understand the dynamic and complex processes that occur inside a combustion chamber. This study concentrates on gas turbine combustors as a whole, and formulates a theoretical design procedure for staged combustors in particular. Not much of literatures available currently in public domain provide intensive study on designing staged combustors. The work covers an extensive study of design methods applied in conventional combustor designs, which includes the reverse flow combustor and the axial flow annular combustors. The knowledge acquired from this study is then applied to develop a theoretical design methodology for double staged (radial and axial) low emission annular combustors. Additionally a model combustor is designed for each type; radial and axial staging using the developed methodology. A prediction of the performance for the model combustors is executed. The main conclusion is that the dimensions of model combustors obtained from the developed design methodology are within the feasibility limits. The comparison between the radially staged and the axially staged combustor has yielded the predicted results such as lower NO_x prediction for the latter and shorter combustor length for the former. The NO_x emission result of the new combustor models are found to be in the range of 50-60ppm. However the predicted NOx results are only very crude and need further detailed study.

C.4.4 Implication of Different Fuel Injector Configurations for Hydrogen Fuelled Micromix Combustors

(ASME Paper No. GT2011-46845)

A design of a hydrogen fuelled micromix concept based combustor is proposed in this paper. The proposed micromix concept based combustor yields improved mixing, which leads to wider flammability range of the hydrogen-air flames compared to conventional kerosene and micromix concept based combustors. This improved mixing allows the combustion zone to operate at a much lower equivalence ratio than the conventional kerosene based and micromix concept based combustors considered in this study. Furthermore, when burning hydrogen the thermal energy radiated to the surroundings is lower (as the result of using lower equivalence ratio) than that of kerosene, consequently resulting in an increased liner life and lower cooling requirement. The aim of this paper is to highlight the impact of using hydrogen as a fuel in gas turbine combustors. It is perceived that this new micromix concept based combustor would also help in achieving low emissions and better performance. Possibilities for lowering NO_x emissions when using hydrogen as a fuel in new designs of micromix combustor are also discussed.

C.4.5 Flame stabilization studies in three backward facing step configuration based microcombustor with premixed methane-air mixtures

(ISABE-2011-1115)

In the present work, experimental investigations on the characterization of flame stabilization behavior in a 2.0 mm base diameter with three backward facing steps and premixed methane-air mixture has been reported. Maximum and minimum diameter in the micro-combustor was kept constant at 2mm and 6mm respectively. Parametric investigations are carried out to understand the effect of change in length of the steps, number of steps, mixture equivalence ratio (ϕ) and flow rate on flame stability limit and wall temperature profiles. It was observed that the recirculation zone created due to sudden expansion at the backward step helps in stabilizing the flame within the micro-combustor and enhances the flame stability limits significantly. The increase in the length of the first and step helps in enhancing the flame stability limits at both lower and higher flow rates. The increase in the length of the third step affects the flame stability limit at higher flow rates only.

C.4.6 Study on Nuclear Energy Powered Novel Air Transport Model and its Feasibility study

(AIAA Paper No. AIAA-2011-6038)

We all know that we have limited resource of petro-chemical fuels which will last for another 40-50 odd years. In comparison to petro-chemical fuels, nuclear fuels does not produce greenhouse gases, therefore it will not contribute to global warming. Substantial literature is available for alternative fuels, which includes hydrogen, bio fuels, electric propulsion, nuclear propulsion etc which are aiming at reducing the emissions and finding an alternative fuel for the future. Out of all the alternative source of propulsion, nuclear seems to be one of the practical ways to travel in future, as other sources would have some limitations which could not be resolved. In the present study, a novel model for air transport has been proposed and its feasibility for air transport has been analyzed. According to the proposed concept all the chemical propelled aircrafts travelling on that sector could be optimized only for takeoff and climb, which would result in increased efficiency. After climbing a certain designated height, these optimized aircrafts would be carried from one location to another by a nuclear powered aircraft. At destination chemical propelled aircrafts could land by their self engines. Optimizing chemical aircrafts for takeoff and climb, and shutting the engine for cruise would lead to less takeoff weight owing to less consumption of fuel, which would eventually help in decreasing the emissions and less fuel consumption.

C.4.7 Hydrogen as a Fuel for Gas Turbine Engines with Novel Micromix Type Combustors

(AIAA Paper No. AIAA-2011-5806)

New design concepts of micromix concept based combustors have been proposed in this study and its feasibility has been checked using computational fluid dynamics. The proposed micromix concept type combustors will help in more rapid mixing of hydrogen with air and will lead to wider flammability range for the hydrogen-air flames as compared to kerosene and other conventional micromix concept based combustors. The designs considered here are aimed at decreasing NOx emissions and therby highlighting the impacts of using hydrogen as a fuel for gas turbine engines. Here two cases are studied and compared with the conventional design and their ability to perform better without any compromise in the output is discussed.

C.4.8 Evaluation of Hydrogen Combustors for Gas Turbines

(10th Annual International Energy Conversion Engineering Conference, AIAA Paper Number AIAA-2012-4227)

Emission targets for aviation are continuously increasing as the impact of pollution and its effect on the global climate are becoming more attentive. Hydrogen has been publicized to be a fuel that can achieve very low emissions being a long term solution as an energy carrier. Although low emissions in hydrogen combustion can only be achieved when burning the fuel at low equivalence ratios with improved mixing. Ideally the fuel would be pre-mixed before combustion, although hydrogen has very high flame speeds with high risks of flashback. This must be considered, whilst burning in a non pre-mixed state prevents these concerns from occurring throughout its applications. When Hydrogen is used for combustion in gas turbines low emissions are acquired using combustor designs such as micro-mix, lean direct injection and highly strained diffusive combustors. Conventional combustors using only hydrogen expel more NO_X emissions than standard configurations, this is due to inadequate mixing and stoichiometric ratios forming around the diffusive flame. Although the addition of Hydrogen to conventional fuels illustrate a considerable improvement in combustive performance. The study of different combustor configurations with Hydrogen are investigated, establishing the properties and issues presented. The combustors studied characterize the advantages of hydrogen combustion increasing flame stability, reducing emissions and promoting increased component life. Hydrogen permits combustion at leaner blow out limits whilst distressing load shedding and relight performance. The purpose of this study is to examine the combustion configurations required to burn Hydrogen effectively exploiting its attributes and potential.

C.4.9 Preliminary Study of a Novel Gas Turbine Combustor Concept based on Hydrogen Synthesis from Kerosene Reformation

(AIAA Paper Number AIAA-2012-4070)

Addition of hydrogen as an additive in gas turbine combustor shows large benefits to the performance of gas turbine engines in addition to reduction in NOx levels. Due to the complexity associated with dual fuel injection, it has not been implemented in gas turbines. In this study a novel combustor has been proposed which would generate hydrogen rich fuel mixture within the combustor, which would eventually aid in combustion process. It is proposed that the novel combustor would have two stages, combustion of ~5% of the hydrocarbon fuel would occur in the first stage at higher equivalence ratios in the presence of a catalyst, which would eventually lead to the formation of hydrogen rich flue gases. In the subsequent stage the hydrogen rich flue gases from the first stage would act as an additive to combustion of the hydrocarbon fuel. Preliminary studies on the proposed combustor have been presented in this paper. It has been preliminary estimated that the mixture of the hydrocarbon fuel and air could subsequently be burned at much lower equivalence ratios than conventional cases, giving better temperature profiles and stability limits. Computational and equilibrium analysis shows reduction in CO, CO_2 with increased hydrogen imput percentage.

C.4.10 A Review of Hydrogen as a Fuel for Future Air Transport

(AIAA Paper Number AIAA-2012-4267)

Innovations in propulsion system have been the key driver for the progress in air transportation and it is expected to grow at a rapid pace. This incurs challenges in aircraft noise reduction and regulation of hazardous emissions. This paper address the issues associated to reduction in hazardous emissions by investigating the properties and traits of hydrogen. Hydrogen as a fuel is most likely to be the energy carrier for the future of aviation due to its potential zero emissions. A historical review has been carried out on hydrogen usage in aerospace industry till today. The challenges of using hydrogen as a fuel for aero applications have been laid down. The paper also shows various strategies analysed in order to evaluate hydrogen's feasibility which includes production, storage, engine configurations and aircraft configurations.

REFERENCES

- **Adkins R. C., Matharu D. S., and Yost J. O., 1981**: "The Hybrid Diffuser", ASME J. Eng. Gas Turbines Power, 103, pp. 229–236, 1981.

- **Adkins R. C., and Yost J. O., 1983**: "A Compact Diffuser System for Annular Combustors", ASME Paper 83-GT-43, 1983.

- **Adkins R.C., 1975(a)**: "A Short Diffuser with Low Pressure Loss", Journal of Fluids Engineering, Vol. 97, pp. 297-302, 1975.

- **Adkins R.C. and Yost J.O., 1979**: "A combined diffuser arrangement", presented at the International Joint Gas Turbine Congress and Exhibition, Haifa, Israel, 1979.

- **Adkins R.C., 1975(b)**:"Tests on a Vortex-Controlled Diffuser Combined with a Pre-Diffuser and Simulated Combustor," Cranfield SME Report No.C1331-D2, Cranfield University, November 1975.

- **An, S. M., Kim, W. S., Lee, S. Y., 2003:** "Spraying of liquid fuel for improvement of reforming performance for hydrogen generation," *International Journal of Hydrogen Energy*, 36, 5342-5349.

- **Badeer, G. H.,** GE Aeroderivative Gas Turbine – Design and Operating Features, GE IAD, GE Power System, Evendale, OH, GER-3695E.

- **Beer J. M., Chigier N. A., 1972**: Combustion Aerodynamics, Applied Science, London.

- **Bingue, J.P., Saveliev, A.V., Kennedy, L. A., 2004:** Optimization of hydrogen production by filtration combustion of methane by oxygen enrichment and depletion*, International Journal of Hydrogen Energy,* 29, 1365-1370.

- **Bingue, J.P., Saveliev, A.V., Fridman, A. A., Kennedy, L. A., 2002:** "Hydrogen production in ultra-rich filtration combustion of methane and hydrogen sulfide," *International Journal of Hydrogen Energy,* 27, 643-649.

- **Blakey S., Rye, L., Wilson, C. W., 2011:** "Aviation gas turbine alternative fuels: A review," *Proceedings of the Combustion Institute*, 33, 2863–2885.

- **Boehman, A. L., Corre, O. L., 2008:** "Combustion of Syngas in InternalCombustionEngines,"*Combustion Science and Technology*, 180 (6), 1193-1206.

- **Bradshaw S. D., 2002**: "Physics-Based, Reduced-Order Combustor Flow Modeling", MSc Thesis, Massachusetts Institute Of Technology, 2002.

- **Breikin, T.V., Herbert, I.D., Kim, S.K., Regunath, S., Hargrave, S.M., Thompson, H.A., and Fleming, P.J. (2006):** 'Staged Combustion Control Design for Aero Engines'. *Control Engineering Practice* 14 (4), 387-396.

- **Burguburu, J., Cabot, G., Renou, B., Boukhalfa, A. M., Cazalens, M., 2011:** "Effects of H_2 enrichment on flame stability and pollutant emissions for a kerosene/air swirled flame with an aeronautical fuel injector," *Proceedings of the Combustion Institute,* 33, 2927–2935.

- **Cheekatamarla, P. K., Finnerty, C. M., 2008:** "Synthesis gas production via catalytic partial oxidation reforming of liquid fuels," *International Journal of Hydrogen Energy*, 33, 5012-5019.

- **Cheng Bo, 2010**: Gas Turbine Technology - Preliminary Design Study of a Low Emission Combustor For Open Rotor Engine, MSc Thesis, Cranfield University, 2010.

- **Cochran, D. L., and Kline, S. J., 1958:** The Use of Short Flat Vanes as a Means for Producing Efficient Wide-Angle Two-Dimensional Subsonic Diffuser, NACA TN 4309.

- **Colban, W., Gratton, A., Thole, K. A., and Haendler, M., 2005:** "Heat Transfer and Film-Cooling Measurements on a Stator Vane with Fan-Shaped Cooling Holes," GT2005-68258.

- **Colin G. and Webster R. C., 1986:** “Cooled Vane”, filed under the patent number 2189553, 25 April 1986.

- **Conrad E.W., 1979**: Turbine engine altitude chamber and flight testing with liquid hydrogen. In: Hydrogen in Air Transportation, International DGLR/DFVLR-Symposium, sequence 20, Germany, 11-14 September 1979.

- **Contreras A., Yigit S., Ozay K., Veziroglu T. N., 1997**: Hydrogen As Aviation Fuel: A Comparison With Hydrocarbon Fuels, Int. J. of Hydrogen Energy, Vol. 22, No. 10/11, pp. 1053-1060.

- **Dahl G. and Suttrop F., 2001 (a)**: “Combustion chamber and Emissions, The Micromix Hydrogen Combustor Technology”, Task Technical Report 4.4-5A, CRYOPLANE Project.

- **Dahl G. and Suttrop F., 2001 (b)**: Combustion chamber and emissions, estimated NOx-reduction potential of hydrogen fuelled aircraft engines, Task Technical Report 4.4-5B, CRYOPLANE Project.

- **Dahl G., Suttrop F., 1998**: Engine control and Low-NOx, combustion for hydrogen fuelled aircraft gas turbines, Int. J Hydrogen Energy, Vol. 23, No. 8, Pp. 695-704, 1998.

- **Danis, A. M., Burrus, D. L., and Mongia H. C., 1997:** “Anchored CCD for Gas Turbine Combustor Design and Data Correlation”, Journal of Engineering for Gas Turbine and Power, Vol.119, P535-545.

- **Demetri, E. P., Topping, R. F. and Wilson, R. P., 1980:** “Study of Research and Development Requirement of Small Gas-Turbine Combustors”, NASA CR-159796.

- **Ekstedt, E. E., Joshi, N. D., Ablett, A. M., 1994:** Triple Annular Combustor for Gas Turbine Engine, U.S. Patent Number 5323604, General Electric Company.

- **El-Shanawany M. S. M. R., Lefebvre A. H., 1980**: Air-Blast Atomisation: The Effect of Linear Scale on Mean Drop Size, J. Energy, vol. 4, no. 4, pp. 184-189, 1980.

- **Frenillot, J. P., Cabot, G, Cazalens, M., Renou, B., Boukhalfa, M. A., 2009:** “Impact of H_2 addition on flame stability and pollutant emissions for an atmospheric kerosene/air swirled flame of laboratory scaled gas turbine,” *International Journal of Hydrogen Energy, 34*, 3930-3944.

- **Fishenden C. R., Stevens S. J., 1977**: The performance of Annular Combustor Dump Diffuser, J. Aircr., Vol. 10, pp. 60-67.

- **Fluent 12.1, 2010**: User Guide. Lebanon, NH.

- **Frazier, T. R., Foglesong, R. E., Coverdill, R. E., Peters, J. E., Lucht, R. P., 2001:** Fuel-Air Mixing and Flame Structure Measurements for Advanced Low Emission Gas Turbine Combustors, Accession Number ADA386682, University of Illinois.

- **GDP Group 1, 2010**: Aero-derivative Industrial Gas Turbine Engine Based on AVIC Team-1 2009 Year’s Core Engine, MSc Thesis, Cranfield University, 2010.

- **Gupta, A. K., Lilley, D. G. and Syred, N., 1984**, “*Swirl Flows*”, Abacus Press.

- **Gobbato, P., Masi, M., Toffolo, A., Lazzaretto, A., 2011:** “Numerical simulation of a hydrogen fuelled gas turbine combustor,” *International Journal of Hydrogen Energy*, 36, 7993-8002.

- **Gohardani, A. S., Doulgeris, G., Singh, R., 2011:** “Challenges of future aircraft propulsion: A review of distributed propulsion technology and its potential application for the all electric commercial aircraft,” *Progress in Aerospace Sciences*, 47, 369–391.

- **Green, J. B., Domingo, N., Storey, J. M. E., Wagner, R. M., Armfield, J. S., Bromberg, L., Cohn, D. R., Rabinovich, A., Alexeev, N., 2000:** “Experimental Evaluation of SI Engine Operation Supplemented by Hydrogen Rich Gas from a Compact Plasma Boosted Reformer,” *Government/Industry Meeting*, Washington, D.C., SAE Paper Number 2000-01-2206.

- **Guo, S. M., Lai, C. C., Jones, T. V., Oldfield, M. L. G., Lock, G. D. and Rawlinson, A. J., 1998:** "The Application of Thin-Film Technology to Measure Turbine-Vane Heat Transfer and Effectiveness in a Film-Cooled, Engine-Simulated Environment," International Journal of Heat and Fluid Flow, vol. 19, pp. 594-600.

- **Haglind F., Hasselrot A., Singh R., 2006**: Potential of reducing the environmental impact of aviation by using hydrogen, The Aeronautical Journal, August 2006, 533-565.

- **Hamelinck, C. N., Faaij, A. P. C., 2001:** “Future prospects for production of methanol and hydrogen from biomass,” *Journal of Power Sources*, 111 (1), 1-22.

- **Hart, D., Freund, P. and Smith, A., 1999:** “Hydrogen – Today and Tomorrow.” In: *Greenhouse Gas R&D Programme*, International Energy Agency (IEA), Cheltenham.

* **Herbert M. V., 1962**: A theoretical analysis of reaction rate controlled systems: Part II, Eighth Symposium on Combustion, The Combustion Institute, 1962, pp 970-982.

* **Heywood J. B. and Mikus T., 1973**: Parameters controlling Nitric Oxide Emissions from Gas Turbine Combustors. AGARD-CPP-125, 1973.

* **Hou, L., Gong, J., Lu, X., 2010:** "Conversion of Emulsified Kerosene in A GasGenerator With Catalytic Reforming," *Combustion Science and Technology*, 182, 1478–1490.

* **Jackson, A., 2009:** "Optimisation of Aero And industrial Gas Turbine design for The Enviourment," *Phd. Thesis*. Cranfield University, Bedfordshire, England, United Kingdom.

* **Jeng, S. M., Flohre, N. M., and Mongia, H., 2004:** "Swirl Cup Modeling-Atomization", AIAA Paper, AIAA 2004-137.

* **Jones R. E., 1978**: Gas Turbine Engine Emissions from Automotive Gas Turbines, ASME paper 78-GT-85, 1978.

* **Joshi, N.D., Epstein, M.J., Durlak, S., Marakovits, S., and Sabla, P.E., 1994:** Development of a Fuel-Air Premixer for Aero-Derivative Dry Low Emissions Combustor, ASME Paper 92-GT-253.

* **Juste, G., 2006:** "Hydrogen injection as additional fuel in gas turbine combustor," *International Journal of Hydrogen Energy,* 31, 2112-2121.

* **Khandelwal, B., Sahota G. P. S., Kumar S., 2010**: "Investigations into the flame stability limits in a backward step micro scale combustor with premixed methane–air mixtures", Journal Micromechanics Microengineering, 20 (9), 095030.

* **Khandelwal, B., Yan, M., Hegde, G., Sethi, V., Singh, R., 2011 (a)**: "Design Procedure of a Reverse Flow Combustor for a Helicopter Engine

with High Temperature Rise," SAE Technical Paper 2011-01-2562, doi:10.4271/2011-01-2562.

- **Khandelwal, B., Hegde, G., Sethi, V., Singh, R., 2012 (a)**: “Design, Evaluation and Performance Analysis of Staged Low Emission Combustor”, ASME Turbo Expo 2012, GT2012-69215.

- **Khandelwal, B., Bao, L., Kumar, K., Sethi, V., Singh, R., 2011 (b):** "Study of Hybrid Diffusers for Use in Gas Turbine Combustors," SAE Technical Paper 2011-01-2496, 2011, doi: 10.4271/2011-01-2496.

- **Khandelwal B., Li. Y., Murthy, P., Sethi, V., Singh, R., 2011 (c)**: “Implication of different fuel injector configurations for hydrogen fuelled micromix combustors,” ASME Turbo Expo 2011, GT2011-46845.

- **Khandelwal, B., Prakash, A., Mahmood, T., Sethi, V., 2011 (d)**: “Study on Nuclear Energy Powered Novel Air Transport Model and its Fesability study,” 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA 2011-6038.

- **Khandelwal, B., 2012 (b):** “Preliminary Study of a Novel Gas Turbine Combustor Concept based on Hydrogen Synthesis from Kerosene Reformation” 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Atlanta, USA, AIAA Paper Number AIAA-2012-4070.

- **Khandelwal, B., Karakurt, A., Sethi, V., Singh, R., 2012 (c):** “A Review of Hydrogen as a Fuel for Future Air Transport” 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Atlanta, USA, AIAA Paper Number AIAA-2012-4267.

- **Khandelwal, B., Karakurt, A., Sethi, V., Singh, R., 2012 (d):** “Study of novel micro-mix combustors to be used in gas turbines; using hydrogen, hydrogen-kerosene, hydrogen-methane and methane as a fuel” 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Atlanta, USA, AIAA Paper Number AIAA-2012-4265.

- **Khandelwal, B., Karakurt, A., Sethi, V., Singh, R., 2012 (e):** "Hydrogen Combustors in Gas Turbines" 10th Annual International Energy Conversion Engineering Conference, Atlanta, USA, AIAA Paper Number AIAA-2012-4227.

- **Khandelwal, B., Zhang, Q., Sethi, V., Singh, R., 2012 (f):** "Preliminary Design and Performance analysis of a Low Emission Aero-Derived Gas Turbine Combustor," The Aeronautical Journal (Submitted for publication).

- **Khandelwal, B., Bao, L., Sethi, V., Singh, R., 2012 (g):** "Computational study on vortex controlled hybrid diffuser" under internal review for publication with an International Journal.

- **Khandelwal, B., Dong, L., Sethi, V., Singh, R., 2012 (h):** "Design and Study on Performance of Axial Swirler for Annular Combustor by changing different design Parameters**" Journal of Power and Propulsion Research** (Submitted for publication)**.**

- **Khandelwal, B., Karakurt, A., Sekran, P., Sethi, V., Singh, R., 2012 (i):** "A review on hydrogen as a future aviation fuel" **Journal of Progress in Aerospace Sciences** (Submitted for publication)**.**

- **Klug, H.G., Bakan, S. And Gayler, V., 1996**: CRYOPLANE – Quantitative comparison of contribution to anthropogenic greenhouse effect of liquid hydrogen aircraft versus conventional kerosene aircraft, EGS XXI General Assembly, Den Haag, 6-10 May 1996.

- **Klug, H.G. And Grassl, H., 1993**: The CRYOPLANE Project, Aircraft using cryogenic fuel and their impact on the atmosphere, European Geophysical Society XVIII General Assembly, Wiesbaden, 3-7 May 1993.

- **Knight M. A., Walker R. B., 1957**: The Component Pressure Losses in Combustion Chambers, Aeronautical Research Council R and M 2987, England.

- **Langston, L.S., Nice, M.L., Hooper, R.M, 1977:** "Three-Dimensional Flow Within a Turbine Cascade Passage," ASME Journal of Engineering for Power, Vol. 102, pp. 21-28.

- **Langston, L.S., 1980:** "Crossflows in a Turbine Cascade Passage," ASME Journal of Engineering for Power, Vol. 102, pp. 866-874.

- **Le Dilosquer M. (1998):** Influence of subsonic aero engine design and flight routes on atmospheric pollution (unpublished PHD thesis), School of Mechanical Engineering, Cranfield University.

- **Leishman, J. G., 2000**: "Principle of Helicopter Aerodynamics", Cambridge University Press, Cambridge, United Kingdom.

- **Lefebvre A. H., 1999** "*Gas Turbine Combustion*", Second edition, Taylor &Francis Group, New York.

- **Lefebvre, A. H., 1989:** "Atomization and Spray, Hemisphere Publishing Corporation".

- **Lefebvre, A. H., 1981**: "Fuels Effects on Gas Turbine Combustion-Liner Temperature, Pattern Factor, and Pollutant Emissions", Journal of Aircraft, Vol 21, No 11, PP. 887-898.

- **Lefebvre A. H., Ballal D. R., 2010**, "*Gas Turbine Combustion: Alternative fuels and Emissions*", Third Edition, CRC Press, Taylor &Francis Group, New York.

- **Lefebvre A. H., 1966**: Theoretical Aspects of Gas Turbine Combustion Performance, Co A Note Aero No. 163, Cranfield, August 1966.

- **Lewis G. D., 1991**: A New Understanding of NOx Formation, Tenth International Symposium on Air-Breathing Engines, ISABE 91-7064, Nottingham, UK, AIAA, Washington DC, pp. 625-9.

- **Leyes II, R. A.and Fleming, W. F., 1999:** "The History of North American Small Gas Turbine Aircraft Engines", American Institute of Aeronautics and Astronautics, Inc., Reston, Virginia.

- **Li Y., 2011**: Numerical Study on Low NOx Hydrogen Micro-mix combustors for Aero Gas Turbines, MSc Thesis, Cranfield University, UK, February 2011.

- **Lilley D. G., Gupta A. K., 2004** "*Combustion Efficiency and the θ-Parameter In the Design and Development of Gas Turbine Combustors*", AIAA Paper, AIAA 2004-3543.

- **Lomax, F. D., Reardon, J., Barbour, J. P., 2002:** "Ststem for hydrogen generation through steam reforming of hydrocarbons and integrated chemical reactor for hydrogen production from hydrocarbon," United States *Patent No. US 6,497,856 B1.*

- **Loukou, A.,Frenzel, I., Klein, J., Trimis, D., 2012:** "Experimental study of hydrogen production and soot particulate matter emissions from methane rich-combustion in inert porous media," *International Journal Of Hydrogen Energy* , 10.1016/j.ijhydene.2012.02.041.

- **Lutz, A. E., Bradshaw, R. W., Keller, J. O., Witmer, D. E., 2003:** "Thermodynamic analysis of hydrogen production by steam reforming," *International Journal of Hydrogen Energy*, 28 (2), 159 – 167.

- **Lynch, S., 2008:** "Flow and Thermal performance of a gas turbine nozzle guide vane with leading edge fillet", Virginia tech.

- **Mahmood, G.I., Gustafson, R., Acharya, S., 2005:** "Experimental Investigation of Flow Structure and Nusselt Number in a Low-Speed

Linear Blade Passage With and Without Leading-Edge Fillets," ASME Journal of Heat Transfer, Vol. 127, pp. 499-512.

- **Marek J. C., Timothy D. S. and Krishna, K., 2005**: Low emission hydrogen combustors for gas turbines using lean direct injection, 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Tucson, Arizona, July 10–13, 2005, AIAA-2005-3776.

- **Matzakos, A. N., Welliongton, S. T., Mikus, T., Ward, J. M., 2004:** "Integrated flameless distributed combustion/steam reforming membrane reactor for hydrogen production and use thereof in zero emissions hybrid power system,"*United States Patent No. US 6,821,501 B2.*

- **Mellor A. M., 1990**: *Design of modern turbine combustors*, Academic Press, London; San Diego.

- **Mellor, A. M. and Fritsky, K.J., 1990 (a):** "Turbine Combustor Preliminary Design Approach", Journal of Propulsion and Power, Vol.6, No.3, May-June 1990, P334-343.

- **Menter F. R., 1974**: "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, vol. 32, pp. 269-289, 1994.

- **Mohammad B. S., Jeng S. M., 2009**: Design procedures and a Developed Computer Code for Preliminary Single Annular Combustor Design, AIAA 2009-5208, 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit.

- **Murthy J. N., 1988**: Gas Turbine Combustor Modelling For Design, Department of Power and Propulsion, Cranfield University, UK.

- **Murthy J. N., 1984**, Combustor Design Program, M.Sc. Thesis, Department of Power and Propulsion, Cranfield University, UK.

- **Murthy, P., Khandelwal, B., Sethi, V., Singh, R., 2011**: "Hydrogen as a Fuel for Gas Turbine Engines with Novel Micromix Type Combustors," 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA 2011- 5086.

- **Nojoumi, H., Dincer., I., Naterer, G. F., 2009:** "Green house emission assessment of hydrogen on kerosene fueled aircraft," *International Journal of Hydrogen Energy*, 34 (3), 1363-1369.

- **Northern Research Engineering, 1974**: Computer Program for the Analysis of Annular Combustors, NASA CR 72374, NASA CR 72375.

- **Nukiyama S., Tanasawa Y., 1939**: Experiments on the Atomization of Liquids in an airstream, Trans. Soc. Mech. Eng. Jpn., vol. 5, pp. 68-75, 1939.

- **Odgers, J., Kretscgmer, D., 1985**: The Prediction of Thermal NOx in Gas Turbine, ASME Paper 85-1GT-126.

- **Oke, R., Simon, T., Shih, T. Zhu, B., Lin, Y. L., Chyu, M., 2001:** "Measurements Over a Film- Cooled, Contoured Endwall with Various Coolant Injection Rates," 2001-GT-140.

- **PAYZER R.J. and RENNINGER S.W., 1979:** Hydrogen fueled high bypass turbofans in subsonic aircraft. In: hydrogen in air transportation, International DGLR/DFVLR-Symposium, Germany, 11-14 September

- **Pedersen-Mjaanes, H., Chan., L., Mastorakos, E., 2005:** "Hydrogen production from rich combustion in porous media," *International Journal of Hydrogen Energy*, 30 (6), 579-592.

- **Pilidis, P., 2010:** Gas Turbine Theory and Performance, Unpublished Lecture Notes, Cranfield University, UK.

- **Pohl H. W., Malychevc V. V., 1997**: Hydrogen in future civil aviation, Int. J. hydrogen Energy Vol. 22, No. 10/11, pp. 1061-1069. 1997.

- **PRATT D.T., ALLWINE K.J. and MALTE, C.P., 1974**: Hydrogen as a Turbojet Engine Fuel – Technological, Economical and Environmental Impact, Proceedings of 2nd International Symposium on Air Breathing Engines, Sheffield, UK, Royal Aero Soc, 1974.

- **Rizkalla A. A., Lefebvre A. H., 1975**: Influence of Liquid Propoerties on Air Blast Atomisation, J. Fluids Eng., vol. 97, no. 3, pp. 316-320.

- **Rizk N. K., 1977**: Studies on Liquid Sheet Disintegration in Airblast Atomisers, Ph.D. Thesis, Cranfield Institute of Technology, 1977.

- **Rizk N. K., Lefebvre A. H., 1980**: Influence of Liquid Film Thickness on Air Blast Atomisation, J. Eng. Power, vol. 102, pp. 706-710.

- **Rizk, N. K., and Mongia, H. C., 1994**: Emission Predictions of Different Gas Turbine Combustors, AIAA Paper 94-0118.

- **Rogers, H. L., Lee, D. S., Raper, D. W., Forster, P. M., Wilson, C. W. and Newton, P. J., 2002:** "The impacts of aviation on the atmosphere," *The Aeronautical Journal,* 160, 521-546.

- **Rokke, N. A., Hustad, J.E., and Berg, S., 1993:** Pollutant Emissions from Gas Fired Turbine Engines in Offshore Practice – Measurements and Scaling, ASME Paper 93-GT-170.

- **Rolls-Royce, 2005**: "*The Jet Engine*", St Ives Westerham Limited, UK, 2005.

- **Roy, R. P., Squires, K. D., Gerendas, M., Song, S., Howe, W. J., and Ansari, A., 2000:** "Flow and Heat Transfer at the Hub Endwall of Inlet Vane Passages-Experiments and Simulations," 2000-GT-198.

- **Rye, L., Wilson, C. W., 2012:** "The influence of alternative fuel composition on gas turbine ignition performance," *Fuel*, 96, 277-283, 2012.

- **Schwerdt, C. 2006:** “Modelling NO_x-Formation in Combustion Processes”, Lund University.
- **Shum F., Moore M., Cohen J., Rosfjord T., Eberius H., Ziemann J., Thomaier D. And Simon B., 1996**: Potential use of hydrogen in air propulsion, Final Report, submitted to Hydro-Québec (contract no. 16233-92-ERE-01500) and European Union (contract no. 4541-91-11 EL ISP PC), 1996.
- **Singh R., 2011**: Gas Turbine Combustors, Lecture Notes, 2011
- **Smith S. F., 1963**: “Improvement in or relating to gas turbine engine combustion equipments”, filed under the patent number 995182, June 20 1963.
- **Sosounov V. and Orlov V., 1990**: Experimental Turbofan Using Liquid Hydrogen and Liquid Natural Gas as Fuel, AIAA 90-2421, Proceedings of AIAA/SAE/ASME/ASEE 26th Joint Propulsion Conference, Orlando, USA, 16-18 July 1990.
- **Stickles R. W., Loveland & et.al, 1993:** “Dilution poles combustor and method”, filed under the patent number 5239818, 31 August 1993.
- **Sutherland G., 1972**:“An investigation into the flow of vortex diffusers”, MSc. Thesis, Cranfield University, 1972.
- **Suttaford P J., 1997:** Preliminary Gas Turbine Combustor Design Using a Network Approach, Department of Power and Propulsion, Cranfield University, UK.
- **Suttarop F., 2001**: DaimlerChrysler Aerospace Airbus GmbH, Method and combustor for combusting hydrogen, 2001, U.S. Pat 6,267,585 B1.
- **Suzuki, T., Iwanami, H., Yoshinari, T., 2000:** “Steam reforming of kerosene on Ru/Al_2O_3 catalyst to yield Hydrogen,” *International Journal of Hydrogen Energy*, 25, 119-126.

- **Svensson F., Hasselrot A., Moldanova J., 2004**: “Reduced environmental impact by lowered cruise altitude for liquid hydrogen-fuelled aircraft”, Aerospace Science and Technology, 8, 307–320.
- **Toledo, M., Bubnovich, V., Saveliev, A. and Kennedy, L., 2009:** “Hydrogen production in ultra rich combustion of hydrocarbon fuels in porous media.” *International Journal of Hydrogen Energy*, 34 (4), 1818-1827.
- **Veziroglu T. N., Babir F., 1998**: ”Hydrogen Energy Technologies”, UNIDO - United Nations Industrial Development Organization, Vienna.
- **Vaught J.M., 1966:** “Turbine stator- combustor structure”, filed under the patent number 3608310, 27 June 1966.
- **Walker A. D., 2002**: “Experimental and Computational Study of Hybrid Diffusers for Gas Turbine Combustors”, Ph.D. thesis, Loughborough University, 2002.
- **Wild, P. J., Verhaak, M. J. F. M., 2000:** “Catalytic production of hydrogen from methanol,” *Catalysis Today*, 60, 3–10.
- **Yan, M, 2010: “**A Reverse Flow Combustor Aerodynamic Design for a Helicopter Engine,” MSc Thesis, Cranfield University.
- **Zess, G.A., Thole, K.A., 2002:** “Computational Design and Experimental Evaluation of Using a Leading Edge Fillet on a Gas Turbine Vane,” ASME Journal of Turbomachinery, Vol. 124, pp. 167-175.
- **Ziemann, J., Shum, F., Moore, M., Kluyskens, D., Thomaier, D., Zarzalis, N. And Eberius, H., 1998(a)**: Low-NO_x Combustors for hydrogen fueled aero engine, *Int J Hydrogen Energy*, 1998, 23, (4), pp 281-288.
- **Ziemann J., Mayr A., Anagnostou A., Suttrop F., Lowe M., Bagheri S. A. and Nitsche Th., 1998(b):** “Potential Use of Hydrogen in Air

Propulsion, EQHHPP, Phase III.0-3”, Final Report, submitted to European Union (contract no. 5077-92-11 EL ISP D).

- **Zhang, Q., 2010:** “Preliminary design of aero-derived gas turbine combustor for dry low emission purpose,” MSc Thesis, Cranfield University.

- **Zhang, L., Baltz, M., Pudupatty, R., and Fox, M., 1999:** "Turbine Nozzle Film-Cooling Study Using the Pressure Sensitive Paint (PSP) Technique," 99-GT-196.

- **Zhang, L., and Pudupatty, R., 2000:** "The Effects of Injection Angle and Hole Exit Shape on Turbine Nozzle Pressure Side Film-cooling," 2000-GT-247.