WARSAW UNIVERSITY OF TECHNOLOGY
Politechnika Warszawska
FACULTY
OF POWER AND AERONAUTICAL ENGINEERING
Wydział Mechaniczny Energetyki i Lotnictwa

DIVISION/*Zakład* **:** Aircraft Engines/ *Silników Lotniczych*

DIPLOMA THESIS
Praca dyplomowa

Master of Science

Nikhil Madan

Examination and analysis of structural changes in the geometry of micro turbine engine components to enhance its performance
Badanie i analiza zmian geometrycznych elementów konstrukcyjnych małego silnika turbinowego w celu zwiększenia jego wydajności

Nr. Index: 257262

Aerospace Engineering

Supervisor/*Promotor***:** prof. nzw. dr hab. inż. Marian Gieras

Warsaw, March 2015
Warszawa, Marzec 2015r.

Statement of the author (authors) of the thesis

Being aware of my legal responsibility, I certify that this diploma:

– has been written by me alone and does not contain any content obtained in a manner inconsistent with the applicable rules,
– had not been previously subject to the procedures for obtaining professional title or degree at the university

Furthermore I declare that this version of the diploma thesis is identical with the electronic version attached.

Statement

I agree / ~~do not agree~~*[1] to make my diploma thesis available to people, who may be interested in it. Access to the thesis will be possible in the premises of faculty library. The thesis availability acceptance does not imply the acceptance of making the copy of it in whole or in parts.
Regardless the lack of agreement the thesis can be viewed by:
- the authorities of Warsaw University of Technology
- Members of The Polish Accreditation Committee
- State officers and other persons entitled, under the relevant laws in force in the Polish Republic, to free access to materials protected by international copyright laws.

Lack of consent does not preclude the control of the thesis by anti-plagiarism system.

Oświadczenie autora (autorów) pracy

Świadom odpowiedzialności prawnej oświadczam, że przedstawiona praca dyplomowa:

- *została napisana przeze mnie samodzielnie i nie zawiera treści uzyskanych w sposób niezgodny z obowiązującymi przepisami;*
- *(nie była wcześniej przedmiotem procedur związanych z uzyskaniem tytułu zawodowego lub stopnia naukowego w wyższej uczelni;*

Oświadczam ponadto, że niniejsza wersja pracy jest identyczna z załączoną wersją elektroniczną.

Oświadczenie

Wyrażam zgodę/ ~~nie wyrażam zgody~~[1] na udostępnienie osobom zainteresowanym mojej pracy dyplomowej. Praca może być udostępniania w pomieszczeniach biblioteki wydziałowej. Zgodna na udostępnienie pracy dyplomowej nie oznacza wyrażenia zgody na jej kopiowanie w całości lub w części.*
Brak zgody nie oznacza ograniczenia dostępu do pracy dyplomowej osób:

- *reprezentujących władze Politechniki Warszawskiej*
- *członków Komisji Akredytacyjnych*
- *funkcjonariuszy służb państwowych i innych osób uprawnionych, na mocy odpowiednich przepisów prawnych obowiązujących na terenie Rzeczypospolitej Polskiej, do swobodnego dostępu do materiałów chronionych międzynarodowymi przepisami o prawach autorskich.*

Brak zgody nie wyklucza także kontroli tekstu pracy dyplomowej w systemie antyplagiatowym.

date	author's (s') signature
data	*podpis autora (autorów) pracy*

KEYWORDS: micro gas turbine, trapped vortex combustion, CFD, navier-stokes equations

(SŁOWA KLUCZOWE)

Abstract

Gas turbine is certainly one of the human inventions of the modern history which has changed the way we perceive our planet today. From wide body aircrafts capable of carrying up to 853 passengers across the planet to the world's largest industrial gas turbine capable of meeting power needs of a present day city like Hamburg, gas turbine technology is one of the promising answers to the growing energy demand with reduced ecological footprint. Over the past few decades, miniature version of this century old technology namely "micro gas turbine" is gaining unprecedented momentum as a viable solution for the quest of sustainable energy systems and a propulsion source for UAV's.

Dependencies of physical laws (fluid, thermal) on system dimensions call for a major research effort to understand the underlying principles and capture the deviation in its operational behavior in micro regime. Proceeding on similar lines, this thesis work is an effort to study these deviations with a special focus on the feasibility of advanced combustion chamber geometries presented in literature for micro gas turbine system. On parallel lines objectives of this study is enhanced system performance in terms of combustion efficiency and reduced NOx emissions.

Key advanced combustion chamber configurations presented in literature for standard gas turbines are categorically studied here and critically assessed for their practical limitations in applicability to a typical micro gas turbine system. To broaden the scope of study, only combustion chamber dimensions comparable to what can be found in combustion chamber of a micro gas turbine are used. No reference to construction, design or operational details of a particular micro gas turbine make or brand is made.

Trapped Vortex Combustion concept is studied in detail as potential combustion chamber geometry for enhanced performance of micro gas turbines using CFD. A self programmed CFD tool written in Python environment is used to solve basic Navier Stokes Equations in 2D for primitive geometry of Trapped Vortex Combustion concept. Anticipated limitations and overall system performance using Trapped Vortex Combustion concept in a micro gas turbine is also presented.

Preface

The presented work is built upon the ground evolved with author's previous work on encapsulating operational behavior of micro gas turbine GTM-120 using classical gas dynamics theory. While comparing the laboratory results of GTM-120 with that obtained from theoretical calculations, this intermediate master project work outlined the significant deviation in combustion dynamics due to miniature dimensions and associated technical limitations of system.

Current study is an effort to take that learning a step ahead and explore further the opportunities, concepts and feasibility of those concepts in redesigning the combustion chamber geometry at micro scale, with overall objective of better combustion efficiency and possibly lower emissions.

This thesis work is systemized in following way. First, a general introduction is provided with prime focus on "micro" class gas turbine engines and its definition. Brief history, current research trends and future of the technology are also presented. Second chapter outlines some general details parts and assembly of any standard gas turbine engine. Chapters 3 to 7 delve into further details on each of these components where thermodynamic principle, classification and features at micro scale are carved out in a step by step manner.

Chapter 8 deals with key advanced configurations of gas turbine combustion chamber reported in literature and are focus of major research efforts amongst academia and industry. Developing further, each section of chapter 8 provides a detailed review of these configurations and presents a critical analysis of their applicability in micro turbine engine.

Computational Fluid Dynamics is the analytical tool used in this study for gaining an understanding of fluid flow processes in Trapped Vortex Combustion (TVC) chamber geometry. Chapter 9 presents CFD studies undertaken for simple back step geometry (used in TVC), defines geometry, influence of geometrical parameters on fluid flow, boundary conditions used and graphical results obtained.

Chapter 9 also outlines the basics of fluid dynamics and discrete mathematics which is written in the form of Python script (program) and introduces the tool itself.

Chapter 10 summarizes this study under heading “Conclusions and Propositions”, where based on conclusions drawn from this study, methods and strategies are proposed which can lead to enhanced performance of micro gas turbine using Trapped Vortex Combustion concept. Chapter 11 leads to closing of this paper with references.

Contents **Page no.**

Nomenclature*

C_p – *specific heat at constant pressure*

C_v - *specific heat at constant volume*

E_t – *total energy*

f – *fuel-air ratio*

M- *mach number*

$\dot{m}_a$ - *mass flow rate for air*

$\dot{m}_f$ - *mass flow rate for fuel*

Q_r – *heat released*

P_a – *ambient pressure*

Re – *Reynolds number*

T_a – *ambient temperature*

u – *velocity component in x- direction*

v- *velocity component in y- direction*

V – *Velocity*

V_0 – *free stream velocity*

w – *velocity component in z- direction*

γ - *adiabatic index*

ρ – *density*

η – *efficiency*

π_c- *compressor pressure ratio*

ϕ - *diameter*

* in SI units, where applicable

1. Introduction

Essentially based on parallel theory of operation, micro gas turbines can be generally seen as miniature version of standard gas turbine engine operating on Joule cycle in open loop configuration. But in contrast to this general perception, the significant deviation in flow and combustion dynamics at scaled down design parameters calls for studying new dimensions in research and limits one to one application of experiences and learning from past 85 years of gas turbine technology.

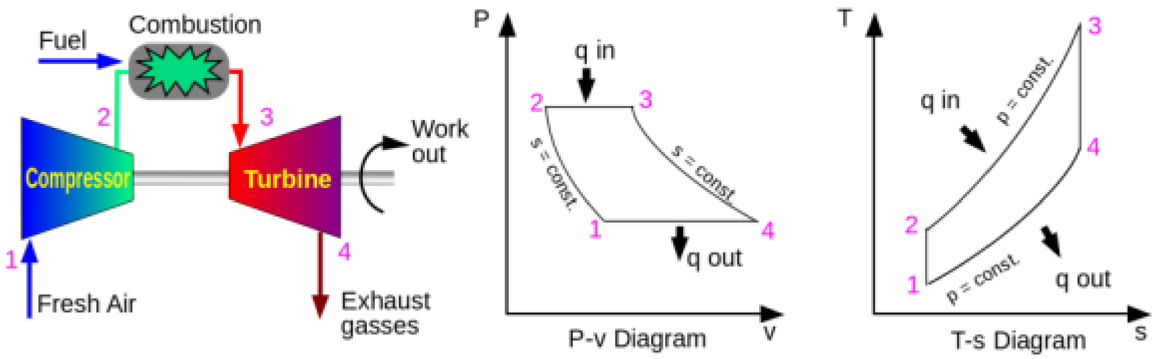


Fig1.1 Thermodynamic cycle of a gas turbine engine (Joule Cycle) [8]

The definition of gas turbine class “Micro” is yet ambiguous so as the market potential of this technology. The size of micro turbine varies substantially. During this study, author have found different ranges like 25 – 80 kW [1], up to 150 kW [1] and 15 – 300kW [2], categorized in “micro” class of gas turbine by different researchers. Similarly, in industry micro gas turbines are available in ranges from 30 - 200 kW (Capstone Turbine Corporation U.S.), 30 – 60 kW (JETPOL Poland), whereas MTT BV from Eindhoven is developing a 3kW system. In this work, gas turbines with power output range from 25 kW to 250 kW are considered and classified in micro class.

Besides potential advantages of a micro gas turbine in combined heat mode like compact size, high power to weight ratio, lower sound levels, better dynamic stability due to less moving parts, the lower efficiencies in standalone configurations still poses a greater barrier in widespread penetration of micro gas turbines into the market. In author’s previous work on capturing operational behavior of micro gas turbine GTM-120 using classical gas dynamics theory, average operational burning efficiency (η_{bo}avg.) of 17.4 % was calculated for operational range of 55000 to 120000 RPM [3]. Substantial changes in aero thermal behavior of gas flows in micro channels and associated technical limitations with micro gas turbine technology leads to lower combustion efficiency with conventional combustion chamber configurations. With ever growing impetus on sustainability, allowable emission norms for all kinds of engine are getting more and more stringent. This trend has pushed researchers and gas turbine manufacturers to come up with new and innovative concepts to meet future regulations on emissions. Many advanced combustion chamber configurations have been proposed over the past few decades for standard gas turbine engines and some of them have been successfully attracting the greater research efforts from industry and academia.

In this work, a step by step analysis of each gas turbine stage at micro level is presented. Some of the key advanced combustion geometries are critically analyzed for their feasibility in micro regime subjected to technical constraints at reduced scale. Drawing the conclusions from this analysis, most promising concept is dealt in preliminary details using CFD. Finally, an overall view for complete micro gas turbine system using this combustion chamber geometry is presented.

2. Common parts of a gas turbine engine

Turbine engines come in a wide variety of shapes and sizes due to many different aircraft mission profiles. However, all gas turbine engines essentially contains following common parts:

- Inlet
- Compressor
- Combustion Chamber/Burner
- Turbine
- Nozzle

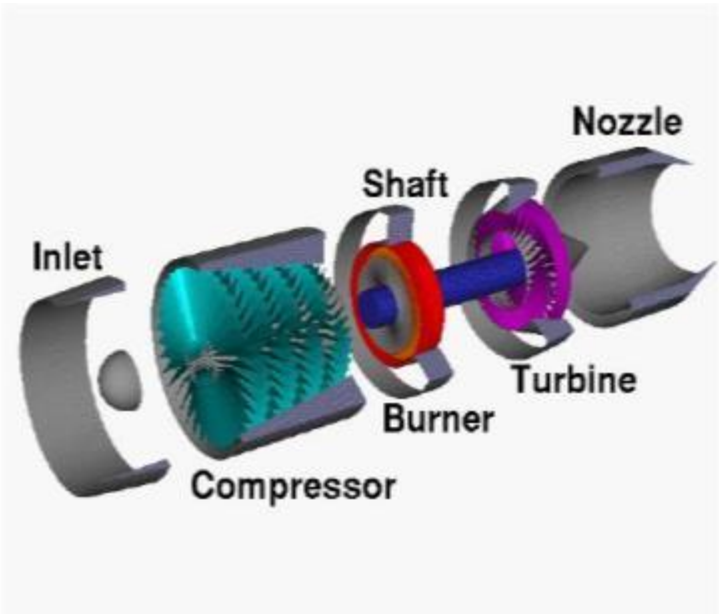


Fig2.1 General Construction of a gas turbine engine [5]

In the exploded view above, in the front of engine towards left is Inlet. At the exit of inlet is compressor which is connected by a shaft to turbine. Compressor and Turbine consists of many rows of specially designed small airfoil blades. Some of the blades are connected with shaft and rotate at very high speed during operation of turbine called rotors, other are stationary set of blades known as stators. The combination of compressor, shaft and turbine is known as Turbomachinery. Between compressor and turbine flow path is combustion chamber or burner, where fuel mixes with compressed air and burned to produce highly pressurized hot gases. These gases then pass through turbine and vented out finally through nozzle. The nozzle performs two important tasks. The nozzle is shaped to accelerate the hot gases to produce thrust. And the nozzle sets through the mass flow through engine. Essentially, micro turbine engine also consists of an inlet, compressor, combustion chamber, turbine and nozzle, but due to technical limitations like ease of manufacturing, material and assembly design of each element can significantly vary to accommodate changed dynamic behavior of gas flows within micro channels.

Subsequent chapters cover details of each element and bring out the important differences and limiting factors in those elements in a micro turbine engine.

3. Inlet

Inlet is a component of aircraft jet engine which captures the incoming propulsive stream tube and conditions it for entrance to engine. The function of the inlet is to adjust the flow from ambient flight conditions to that required for entry into fan or compressor of the engine. It must perform this function over full range of speed from static (take-off) to highest Mach number. For any inlet there are two major requirements:

- to bring the inlet flow to the engine with highest possible stagnation pressure, this is measured by Inlet Pressure Recovery
- to provide the required engine mass flow

Since inlet does no thermodynamic work, therefore the total temperature through the inlet is constant. Referring to Fig1.1 above, $T_2 = T_0$ or $T_2/T_1 = 1.0$, where 0 is the ambient state. The inlet pressure recovery is given by p_2/p_0, depends on a wide variety of factors, including the shape of inlet, the speed of the aircraft, the airflow demands of the engine and aircraft manoeuvres.

Recovery losses associated with boundary layer on the inlet surface and or flow separation in the duct are included in the inlet efficiency factor, $\eta_i = p_2/p_1$. For subsonic flight speeds these losses are the only losses but for supersonic speed additional losses due to shock waves are also significant to take into account.

For military specifications,

$$\text{Mil. Spec., } M < 1.0 : p_2/p_1 = \eta_i \times 1.0$$

&

$$\text{Mil. Spec., } M > 1.0 : \frac{p_2}{p_1} = \eta_i \times \left[1 - 0.075\,(M-1)1.35\right]$$

There is an additional inlet performance parameter called spillage drag. Spillage drag occurs when the inlet "spills" the air around the outside instead of conducting the air into the compressor face. The amount of air that goes through the inlet is set by the engine and depends on flight altitude and throttle settings. The inlet is usually sized to allow maximum airflow that the engine can ever demand. Also as the flow is brought from free stream conditions to the compressor face, the flow may be distorted by the inlet.

At compressor face one portion of the flow may have a higher velocity or pressure than another. The flow may be swirling, or some section of the boundary layer may be thicker than the other due to inlet shape. As the compressor blades face distorted inlet flow, the rapidly changing flow conditions may result into flow separation and a compressor stall which can cause structural problems for the compressor blades. A good inlet must produce high pressure recovery, low spillage drag and low distortion.

3.1 Thermodynamics of inlet

Thermodynamically speaking, inlet does zero work on the flow in an ideal case where the intake process is assumed to be isentropic process, hence pressure and temperature at the face of compressor is represented by simple equation for an isentropic process as,

$$P_{02} = P_{01} = P_a(1 + \frac{\gamma - 1}{2} M^2)^{\frac{\gamma}{\gamma - 1}}$$

and stagnation temperature is given as,

$$T_{02} = T_{01} = T_a(1 + \frac{\gamma - 1}{2} M^2)$$

Whereas in practical situation there are many losses associated with flow mainly due to skin friction and shocks (in supersonic inlets), hence making intake process as adiabatic but irreversible. To account for various losses, above equations are treated with additional efficiency factors to design inlet with sufficient approximations.

The pressure ratio within the inlet may be provided and outlet pressure is obtained from relation:

$$r_d = \frac{P_{02}}{P_{0a}}$$

Alternatively, inlet or diffuser efficiency (η_d) is taken into consideration. Then outlet pressure is given by

$$P_{02} = P_a \left(1 + \eta_d \frac{\gamma_c - 1}{2} M^2{}_a\right)^{\frac{\gamma_c}{(\gamma_c - 1)}}$$

$$T_{02} = T_a\left(1 + \frac{\gamma_c - 1}{2} M^2{}_a\right) = T_{0a}$$

Temperature is calculated using the similar equation as in ideal case since the outlet temperature is independent from the losses. Strongly depending upon flight Mach number, diffuser efficiency, η_d varies between $0.7 < \eta_d < 0.9$ for a standard gas turbine.

3.2 Classification of gas turbine inlets

Inlet of aircraft jet engines come in variety of shapes and sizes which are usually classified on the basis of speed of the aircraft. On this criterion, general classification of inlets is:

- Subsonic Inlets
- Supersonic Inlets
- Hypersonic Inlets

Subsonic Inlets: For subsonic aircrafts like large airliners, a simple, straight, short inlet works quite well. On a typical subsonic inlet, the surface of the inlet from outside to inside is continuously smooth curve with some thickness from inside to outside. The foremost part of the inlet is called as highlight or the inlet lip. A subsonic aircraft has inlet with relatively thick lip.

Fig3.1. Subsonic inlet [5]

Supersonic Inlets: Inlets for supersonic aircraft on other hand have relatively sharper lip. The inlet lip is sharpened to minimize the performance losses due to shock waves at supersonic speeds. For a supersonic aircraft, the inlet must slow down the flow to subsonic speeds before it reaches compressor.

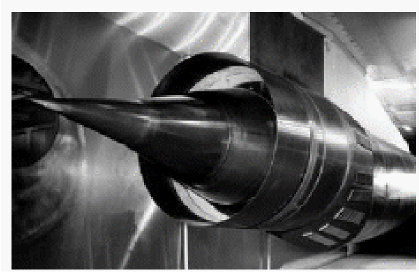


Fig3.2. Supersonic inlet [5]

Hypersonic Inlets: Inlets for hypersonic aircrafts presents the ultimate design challenge. For ramjet powered aircraft the inlet must bring the high speed external flow down to subsonic regime in burner. High stagnation temperatures are dominant in such flight speed conditions. For scram jet powered aircraft the heat conditions are even worse because of higher flight Mach number. Scramjet inlets are usually highly integrated into the fuselage of the aircraft.

Fig3.3. Fuselage integrated hypersonic inlet [5]

3.3 Inlet design for standard gas turbine engines in civil and military aircrafts

In subsonic civil aircraft engine, various aspects have to be considered when designing an inlet. The fundamental problem is to define lip thickness and profile, which has to satisfy two opposing conditions:

- The high Mach number cruise condition, which require a thin leading edge to avoid high external supersonic overspeed and drag.
- The take-off condition, which requires a thick lip to avoid an internal separation due to a too long high overspeed of the entry flow as it turns round the lip.

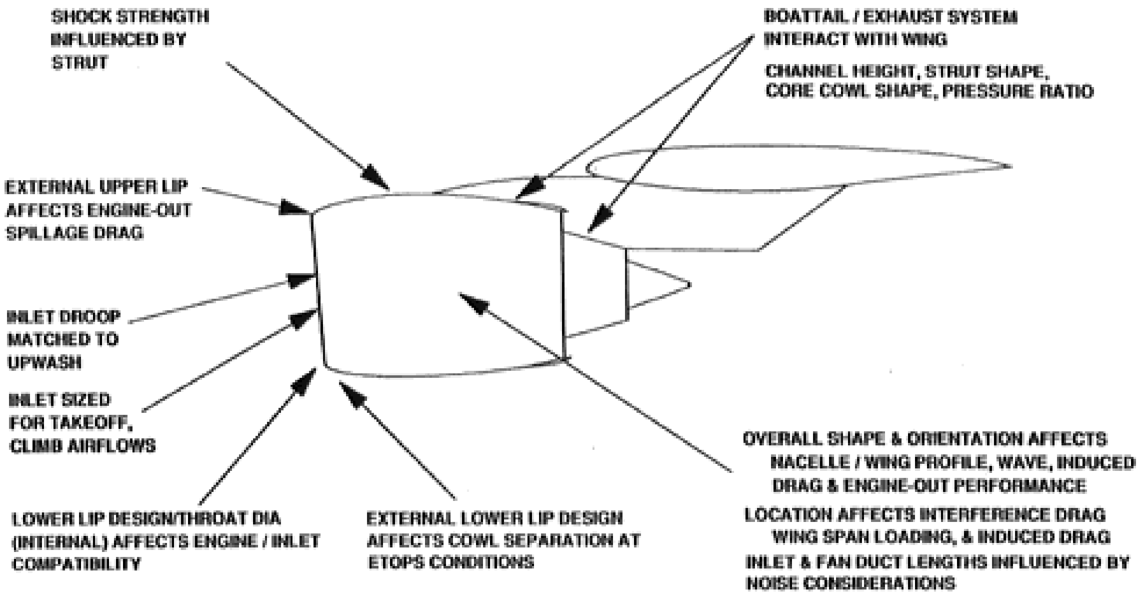


Fig3.4. Aerodynamic design parameters of a subsonic inlet of civil aircraft

Subsonic fixed geometry inlets which are commonly seem in civil aircrafts are always a compromise between various conflicting conditions, major being the different air mass flow requirements at different speed regimes. At take-off & zero aircraft velocity, the engine operates in air sucking mode and due to maximum thrust requirement for take-off, mass flow required is very high as compared to that required when aircraft is in cruise mode due to ram compression. Therefore to meet the different mass flow requirements over flight profile, many difficulties are encountered by inlet designer to deliver an optimum design.

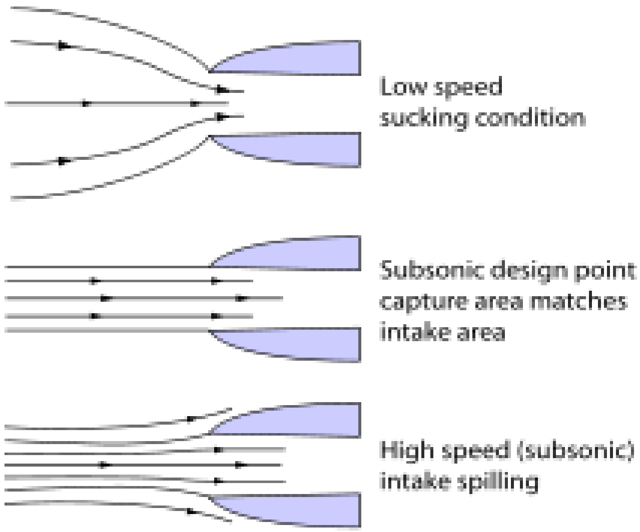


Fig3.5. Subsonic inlet is a tradeoff between mass flow rates at various flight conditions.

At take -off there is a special requirement, which is to avoid large drag rise in the case of engine failure during second segment of the take-off trajectory. This requires a larger external lip radius.

Another important condition which is to be considered while designing an inlet for large aircraft engines is cross-wind. In this case, an internal flow separation may occur, either at low RPM ,due to internal pressure gradient and low Reynolds number or at high RPM due to high overspeed of the flow turning around the lip facing the cross wind. In some cases, separation can only exist in static conditions and as the aircraft pickup a certain speed on runway, it rapidly disappears.

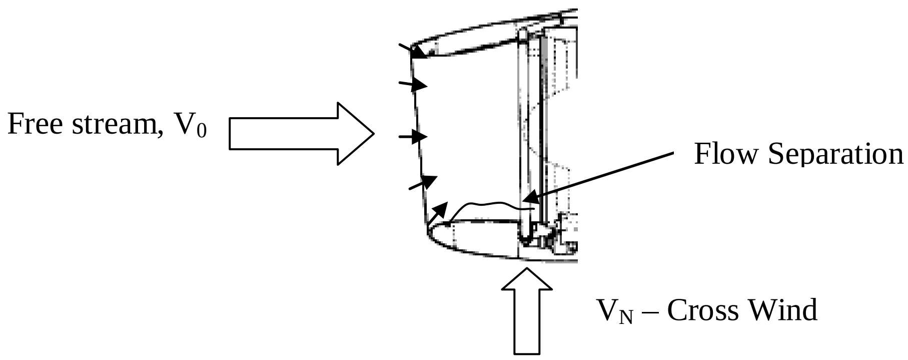


Fig3.6. Internal flow separation due to cross wind

The subsonic inlets are usually equipped with variable geometry devices in order to avoid flow separation and restore good performance at low speed. Auxiliary doors are the simplest of such devices but they introduce large acoustic problems and inlet noise. To avoid such noise internal control of the lip is applied at low speed by blowing or sucking.

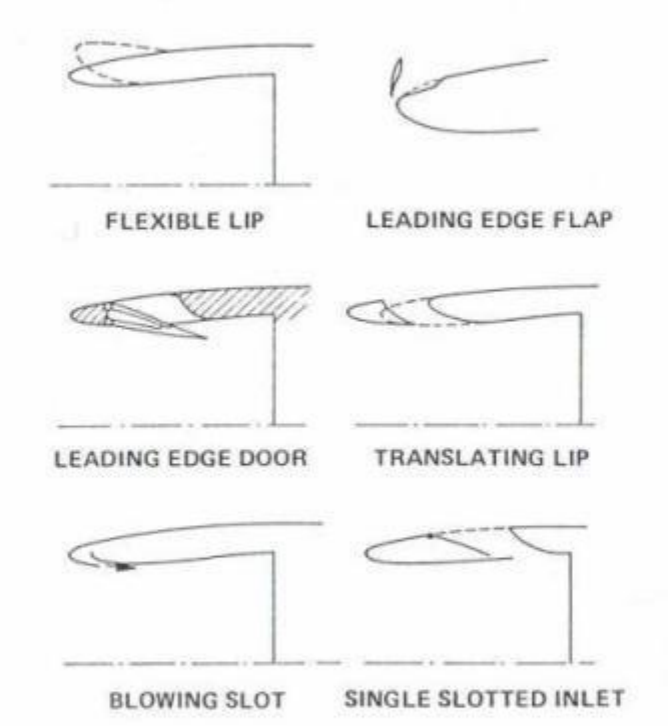


Fig3.7. Variable inlet geometry devices to avoid flow separation

The design choice and size of intakes for military aircraft and especially fighter aircraft is based on performance requirements, but has to be compromise depending upon the relative importance of operation at subsonic or supersonic speeds.

- engine flow demand at high subsonic speeds at altitude generally determines the maximum entry area
- variable compression surface is necessary if the high drag of spilled flow at supersonic speeds is to be avoided and benefit taken of pre-compression of the entry flow by an efficient shock pattern

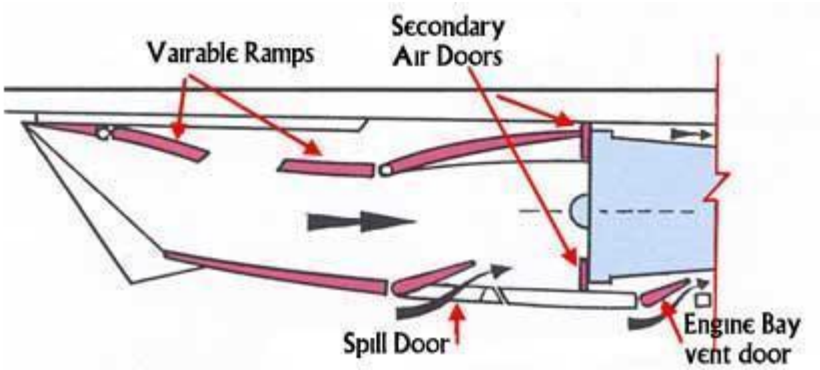


Fig3.8. Variable compression inlet surfaces (ramps) used in military fighter aircrafts

- location of intakes at subsonic speeds plays an important role in having low total pressure distortion and minimum swirl
- the detrimental effects of forebody boundary layer are generally to be avoided and local flow angularity, which can be quite extreme at high angle of attacks is a particularly important parameter for modern highly manoeuvrable aircraft
- inlet of a supersonic military jet has a relatively sharp lip. The inlet lip is sharpened to minimize the performance losses from shock waves that occur during supersonic flight. A supersonic inlet must bring the flow velocity to subsonic range before it enters compressor. Earlier supersonic aircrafts used a central cone configuration as in famous SR-71 blackbird and latest aircrafts are using variable geometry inlet as shown in fig.3.8 above.

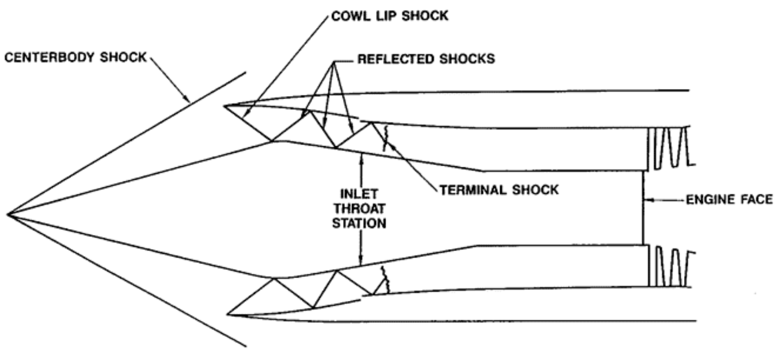


Fig3.9. Central cone used in supersonic aircraft to produce multiple weak shocks

- supersonic inlet is designed to produce multiple weak shocks before the inlet air enters the compressor. Multiple weak shocks (minimum 4-5 shocks) are always ensured by inlet design over the full mission profile in supersonic range to give the optimum performance.

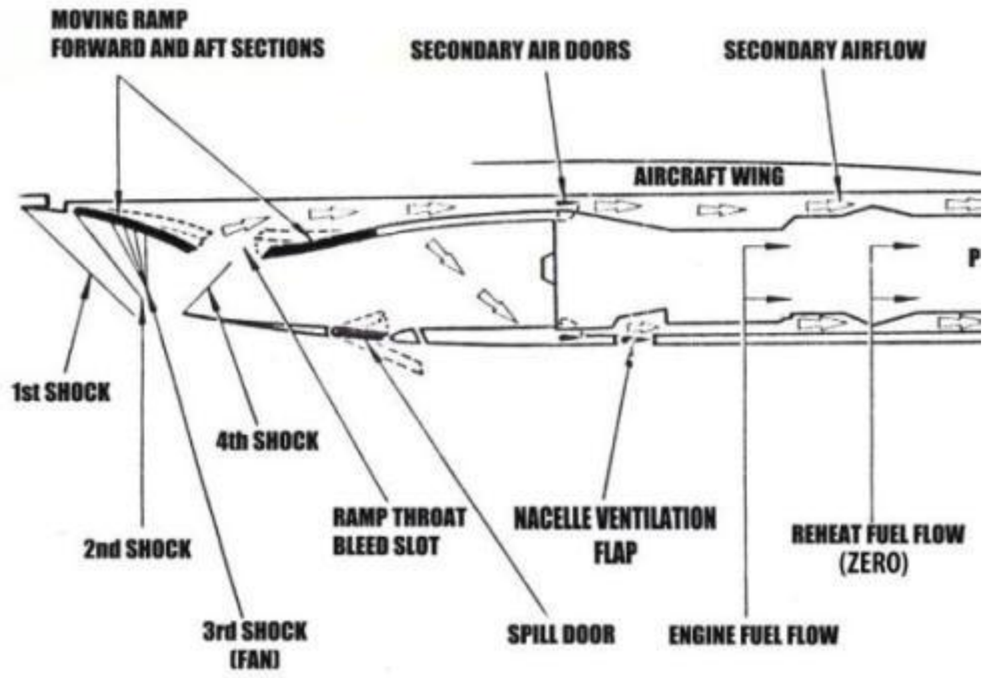


Fig3.10. Multiple weak shocks generated using inlet ramps in F-16

3.4 Inlets for micro jet engines:

Inlets for micro turbine jet engines are predominantly Pitot type inlets specifically designed for centrifugal compressors. The major design challenge for inlets in micro jet engines is to minimize the distortion of inducted air and boundary layer problem.

- micro turbine engines are commonly using electric starter motor which is placed in front of inlet and coupled with compressor shaft. Starter motor and fixing elements are not only creating a drag but also distort the inlet air to high degree. The diameter ratio of motor assembly to the inlet diameter have strong influence over the quality of inlet air in terms of flow stability, therefore this diameter ratio is required to be optimised

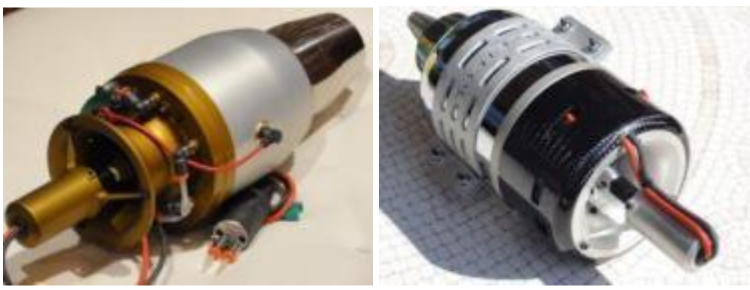


Fig3.11 Micro turbine engines Jetpol GTM-120 & GTM-85 with mounted starter motor [6]

- due to high complexity in miniaturization of an axial compressor, centrifugal compressors have gained a strong foothold in micro jet turbines. To gain better compressor efficiency, inlets of micro turbine engine are usually designed in snail shape to ensure low pressure losses during compression process.

Fig3.12. Centrifugal compressor and specifically designed inlet assembly of GTM-120[6]

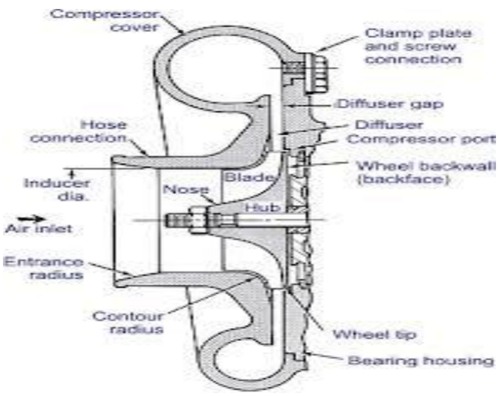
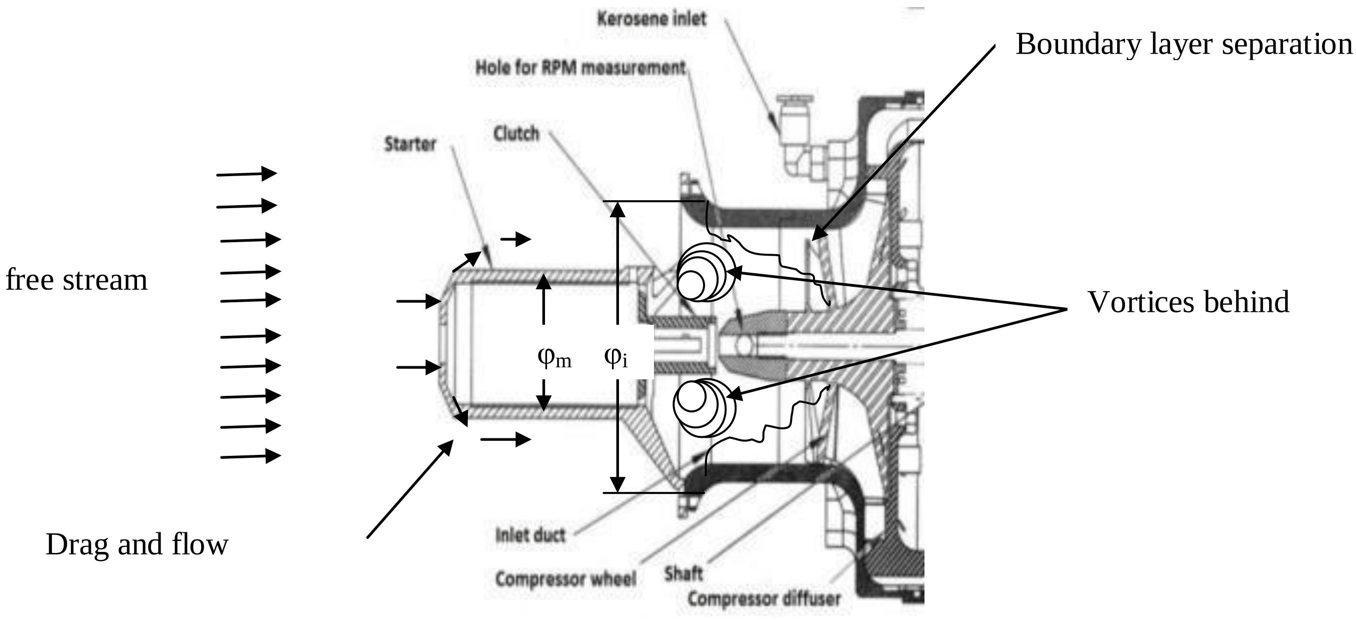


Fig3.13 Cut section of a micro jet engine showing snail shape inlet

- one of the major limiting factor in obtaining the high compression efficiency in micro turbine engine is the small diameter of inlet. With increasing velocity of engine and decreasing diameter of inlet the growing boundary layer effect results in decreasing compressor efficiency which may lead to flame instability in combustion chamber due to insufficient air. During static conditions this situation can lead to complete engine blow off but during flight conditions the ram effect may compensate but still with reduced thrust output

- the area behind the motor assembly in micro turbo jet engine is always susceptible to vortices which must be taken well into account for engine performance



(φ_i / φ_m must be optimized)

Fig3.14 Drag and flow distortion due to starter assembly and flow separation in inlet

4. Compressor

All gas turbine engines are mounted with a compressor to compress the sucked air to an optimum pressure and feeding it to combustion chamber. Compressor performances have a large influence on overall performance of a gas turbine engine. Principally, there are two general kinds of compressors appear in a gas turbine assembly: Axial and Centrifugal. In fig. 4.1 below, both types are shown with compressor position in gas turbine assembly.

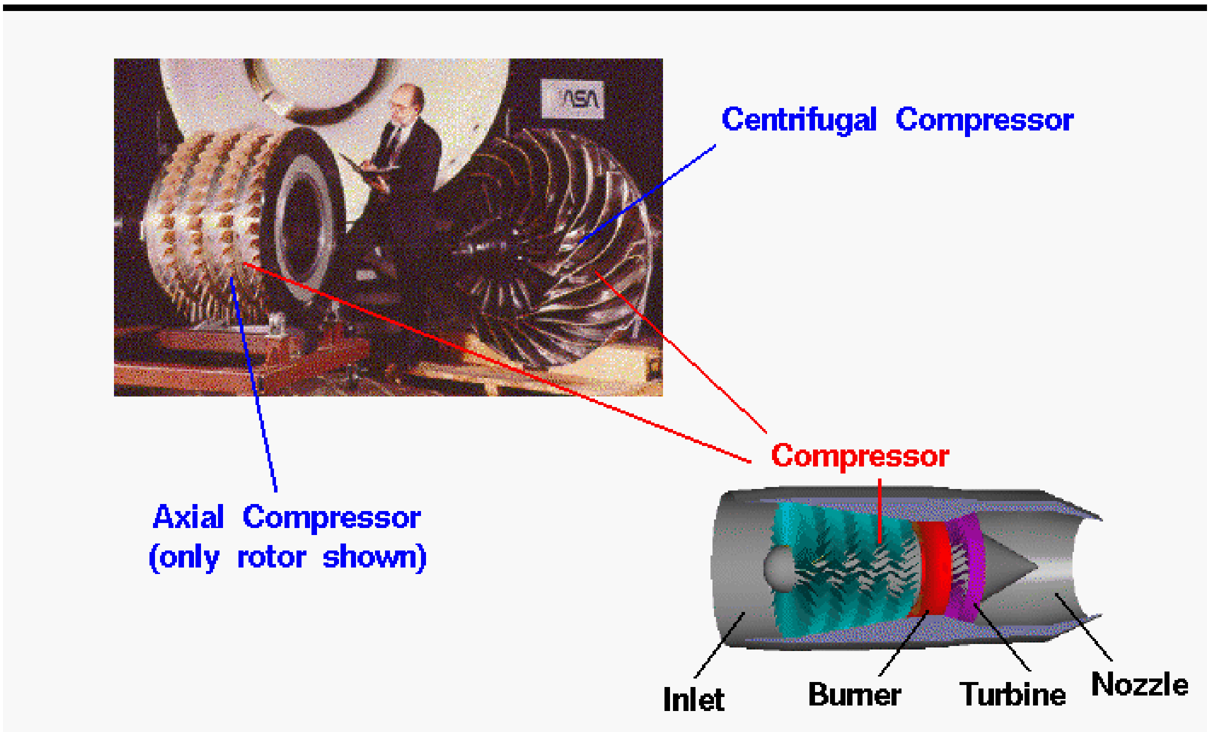


Fig4.1. Axial and Centrifugal compressor with position in gas turbine assembly [5]

In axial compressor, flow through the compressor travels in a direction parallel to its axis, whereas in centrifugal compressor as shown, the flow enters at the centre of hub and leaves the compressor in a radial direction. Centrifugal compressors, which were used in earlier era of jet engines as still find use in small turbojets and turboshaft engines and as pumps on rockets. Modern large turbojet and turbofan engines use axial compressors.

It is interesting to note that even though an average, single stage, centrifugal compressor can increase the pressure by a factor of 4, in comparison to a factor of 1.2 of a similar single stage axial compressor; axial compressors are predominant in modern gas turbines. The important factor responsible for this change is the ease of coupling axial compressors to produce a multistage axial compressor. It is much more difficult to produce an efficient multistage centrifugal compressor because flow has to be ducted again back to the centre of hub. Also, since flow is turned perpendicular to the axis, an engine with centrifugal compressor is usually larger in diameter thus giving rise to an additional aerodynamic drag.

4.1 Thermodynamics of compressor

In ideal operation compression process is assumed to be isentropic and pressure ratio,π_c is usually known. Pressure and temperature outlet of the compressor in ideal case are simply evaluated by isentropic temperature –pressure relation as follows,

$$P_{03} = (P_{02})(\pi_c)$$

and

$$T_{03} = T_{02}\left[\frac{P_{03}}{P_{02}}\right]^{\frac{(\gamma-1)}{\gamma}}$$

In real case, compression process is treated as irreversible hence compression efficiency,η_c is employed to calculate the outlet temperature as shown below,

$$T_{03} = T_{02}\left[1 + \frac{\pi_c^{\frac{(\gamma_c-1)}{\gamma_c}} - 1}{\eta_c}\right]$$

Compressor efficiency, η_c varies between 0.85 < η_c< 0.90 for standard gas turbines

4.2 Types of compressors

As discussed previously, compressors are classified into axial and centrifugal type compressors. Details on each type are elaborated in this section.

4.2.1 Axial Compressor: In axial compressor, the air flows parallel to the axis of rotation. This compressor contains several rows of airfoil cascades. There are two kinds of airfoil sets, one which are mounted on the central shaft and rotate at very high speeds called as Rotors. Other set of airfoils, which are stationary are called as Stators are connected to upper casing of assembly and act as guide vanes to smoothly channelize the airflow to the next set of rotating blades or rotors. As shown in fig. 4.2 below, rotors and stators are generally arranged in alternative sets in axial compressor stage.

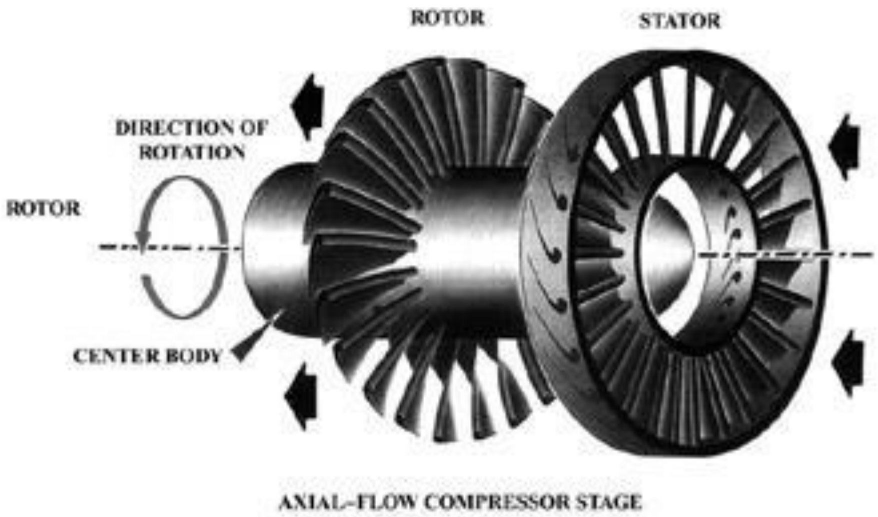


Fig.4.2 Axial compressor stage assembly with alternative rotor, stator set [23]

Each blade on a rotor or stator introduces a pressure variation much like an airfoil or a propeller. But in contrast, the blades of an axial compressor are much closer, therefore significantly changing the flow dynamics around the blade. Compressor blades continuously pass through the wakes of upstream blades that generates unsteady flow. CPR or compressor pressure ratio, shaft RPM to achieve that CPR and efficiency factor are main performance parameters of a compressor.

4.2.2 Centrifugal Compressor: In a centrifugal compressor, the flow is not only compressed by bringing in pressure variations like by airfoils but also accelerating the fluid by changing its direction. Although the inlet flow is more or less axial but the rotor exit flow is in radial direction which is slowed down to recover static pressure by using a diffuser. The diffuser also restores the flow direction to axial for the next process which can be either a next compression stage or a combustion chamber. Similar parameters like CPR, shaft RPM and efficiency are used to characterize performance of centrifugal compressor. Fig. 4.3 below shows centrifugal compressor stage assembly.

Fig.4.3 Centrifugal compressor stage assembly [23]

4.3 Compressor in micro gas turbines

Due to simplicity in design and fabrication, compactness, lower cost and lower weight, centrifugal compressors have gained an omnipresent place in micro gas turbine technology. On the other hand, very design of axial compressors limits their efficient use below certain dimensional scales. With decreasing dimensions in axial compressors, the airfoil in the rear stages are not rugged and thin like razor blades, thus making it unsuitable to stand high turbine RPM's of about 30,000. Whereas impeller of centrifugal compressor is manufactured from metal of relatively higher thickness which can easily withstand high centrifugal stresses generated due to RPM of 30,000 or more. Fig. 4.4 below shows centrifugal compressor assembly in a micro gas turbine.

Fig 4.4 Centrifugal compressor with diffuser in a micro gas turbine [22]

Another limitation of an axial compressor design is that with decreasing scale of gas turbine, distance between airfoils/blades may decrease significantly below the limit to produce the required level of compression or zero effective compression.

5. Combustor Chamber

In all available types of modern day gas turbine engines, apart from different construction details there are some common parts. All turbine engines without limitation of size, power and performance have a combustion chamber or burner, in which fuel is burned in presence of high pressure air fed from compressor.

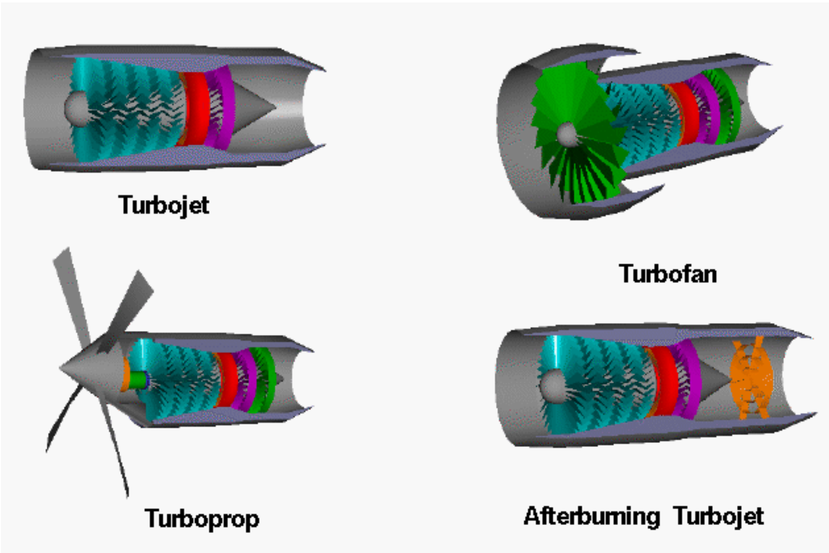


Fig5.1 Different types of gas turbine engines with common parts (burner in red) [5]

Combustion chambers are also used in supersonic engines like ramjet and scramjet. Though intrinsic design vary from that of a gas turbine engine but thermodynamic principle of operation is same. Combustion chamber sits in between compressor and turbine to deliver the resulting gases to the turbine at a uniform temperature as shown in figure below.

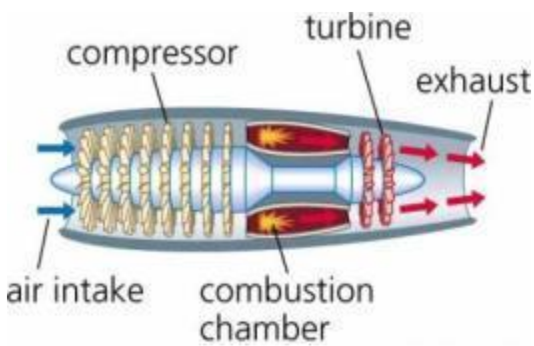


Fig5.2. Basic gas turbine assembly with gas flows

5.1 Thermodynamics of combustion chamber

For single spool engine, the outlet conditions of compressor are treated as inlet conditions at combustion chamber. In ideal scenario, zero pressure drop is assumed, hence pressure at inlet and outlet of combustion chamber remains constant,

$$P_{03} = P_{04}$$

Temperature at the outlet of the turbine is determined from the metallurgical limits of the turbine blade material, also known as Turbine Inlet Temperature (TIT). Mass flow rate of the burnt fuel is calculated from the energy balance of the combustion chamber.

$$(\dot{m}_a + \dot{m}_f)C_{ph}T_{04} = \dot{m}_a C_{pc} T_{03} + \dot{m}_f Q_R$$

$$\frac{\dot{m}_f}{\dot{m}_a} = f = \frac{C_{ph}T_{04} - C_{pc}T_{03}}{Q_R - C_{ph}T_{04}}$$

In real situation, due to fluid friction pressure drop results in lower pressure at the outlet of combustion chamber than its value at inlet. Therefore, pressure at combustion chamber outlet is either given in percentage decrease or absolute pressure drop ΔP_{cc},

$$P_{04} = P_{03} - \Delta P_{cc}$$

or

$$P_{04} = P_{03}(1 - \Delta P_{cc}\%)$$

Fuel-to-air ratio is now calculated by taking into consideration burner efficiency, η_b using relation,

$$\frac{\dot{m}_f}{\dot{m}_a} = f = \frac{C_{ph}T_{04} - C_{pc}T_{03}}{\eta_b Q_R - C_{ph}T_{04}}$$

Burner efficiency, η_b varies between $0.97 < \eta_b \leq 0.99$ for standard gas turbine engine.

5.2 Design and working principle

The most important design factor in any combustion chamber is to ensure that exit gas temperature must not exceed the allowable temperature limits of turbine. A schematic of combustion chamber is shown in Fig.2.3. About 50% of the air entering into combustion chamber mixes with fuel and supports its combustion. The remaining air – bleed air or secondary air is simply used to carry away products of combustion and cooling the burner surfaces.

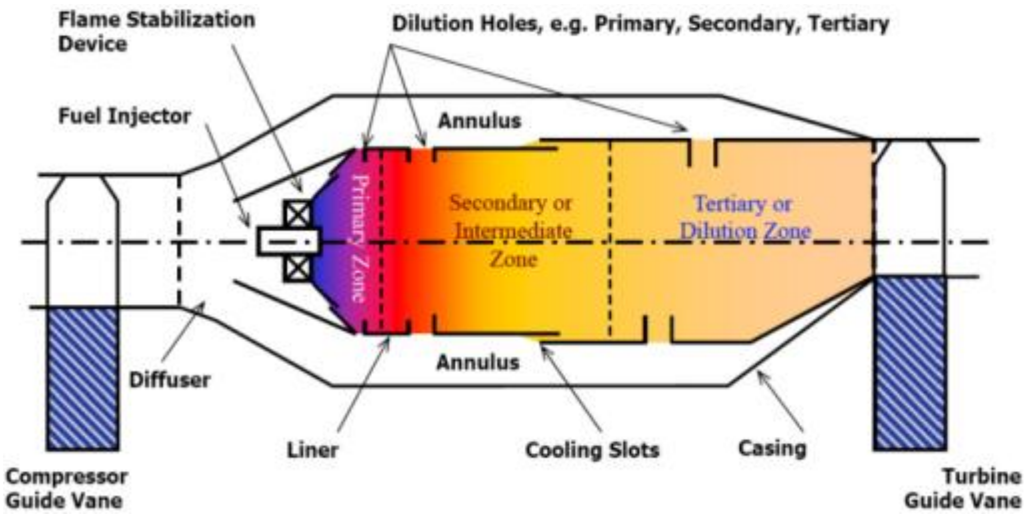


Fig5.3 Schematic of a combustion chamber [2]

The ratio of total air to fuel varies among the different engine types ranging from 30 to 60 parts of air to 1 part of fuel by weight. In modern engine designs only 15 parts out of average 40:1 contribute in burning process to maintain certain limits at given pressure for stable combustion to occur.

5.3 Construction

Driven by material requirement to withstand high temperature gases released during combustion process, combustion chambers are traditionally constructed with high strength nickel based super alloys. Hastelloy -X and Nimonic- 263 are some of the legacy materials for high temperature applications like in combustion chamber. Cobalt based super alloy like HA-188 is finding relatively new place as construction material for gas turbine combustion chambers.

Combustion chamber material choice is not only an important factor from high temperature strength point of view but also from material degradation, corrosion and heat loss point of view. Prevention of heat loss and corrosion of material are simultaneously realized in standard gas turbine by using Thermal Barrier Coatings or TBC's. Yttria stabilized Zirconia as popularly known in industry or $ZrO_2 - Y_2O_3$, finds a commonplace as TBC and coated on combustion chamber parts using plasma spray technique. Thermal Barrier Coatings have played an important role for improved efficiencies of combustion process in gas turbines mainly by preventing significant heat loss. To meet further demands on increased efficiencies and reduced emissions to comply with future regulations, new TBC materials are constantly being researched in industry and academia.

Conventional combustion chamber in gas turbines is usually an annular assembly with cylindrical outer casing and perforated inner liner. Outer casing and inner liner assembly forms a channel to distribute incoming compressed air into two streams and perforations in liner feeds the primary, secondary and tertiary air bleeds into combustion chamber at proper axial location.

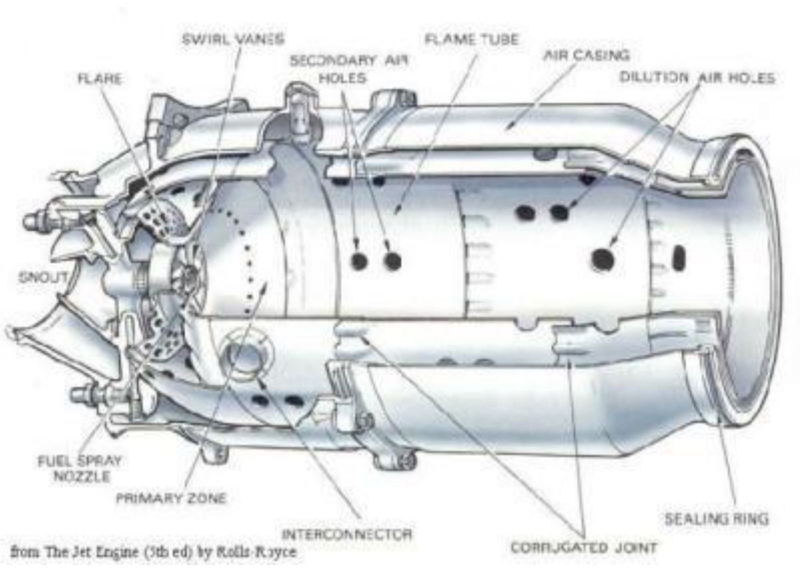


Fig5.4 Cut-section view of a combustor in conventional combustion chamber [21]

Other elements of combustor includes fuel spray nozzle, swirl vanes and flame tube. Fig. 5.4 presents a detailed cut section of view of conventional combustion chamber used in Rolls-Royce jet engine. Swirl vanes are auxiliary components which are added to introduce vortices, hence to improvise mixing of fuel and compressed air.

5.4 Assembly and configuration

Combustion chambers are usually constructed by assembling series of combustors. These configurations can be annular type, can type or can- annular type. Fig 2.5 below shows these configurations in detail.

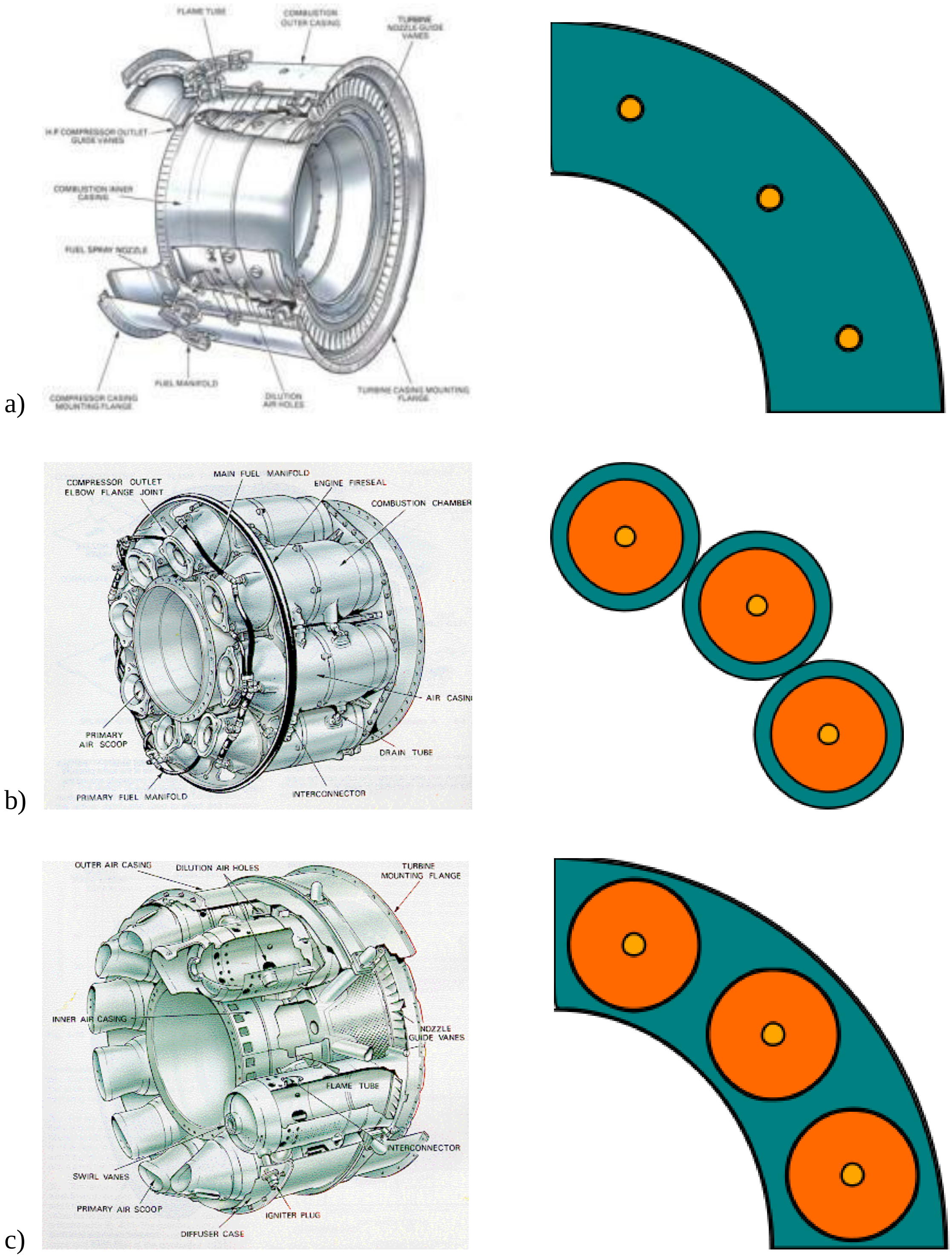


Fig5.5 Types of combustion chambers: a) Annular, b) Can, c) Can-annular [20] [8]

5.5 Combustor Chamber for Micro Gas Turbine

Combustion chambers in micro turbine are essentially working on the similar thermodynamic and operational principles as that of a standard gas turbine. Due to ease of construction at smaller dimensions, most of the micro turbine manufacturers predominately employ annular type configuration. In a relatively simple perspective combustion chamber at micro scale can been somewhat similar in geometry to single burner or combustor can in standard gas turbine. Essential modifications to accommodate shaft connecting compressor and turbine are adopted in design. Fig. 5.6 shows a cut section of micro turbine engine with annular type combustion chamber, with outer casing, perforated liner and innermost casing to accommodate connecting shaft.

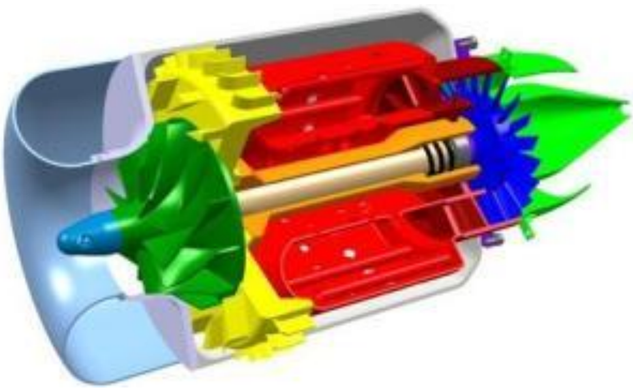


Fig5.6 Cut section of micro gas turbine with annular combustion chamber in red [19]

Due to lower range of gas temperatures and large heat loss rate in micro combustion chamber, conventional materials sufficiently meet the demands of strength well within allowable temperature limits. Lower temperatures also terminate the scope of Thermal Barrier Coatings in micro gas turbines thus leading to additional weight and cost savings.

Fig5.7 Micro turbine combustion chamber parts and assembly [6]

5.6 The Combustion Process

5.6.1 Types of Flames: In literature flames in a combustion chamber of a gas turbine engine is generally classified as diffusion, pre-mixed and partially pre-mixed. If air and fuel are mixed prior to injection in combustion chamber pre-mixed flame will propagate from heat source to the unburnt mixture in reaction zone. In diffusion type flame, fuel and oxidizer are injected right into the place and same time where reaction takes place.

In diffusion type process basically employs turbulence to provide numerous sites in mixture with stochiometric ratio, therefore giving a stable flame front, hence been traditionally preferred by many gas turbine manufactures. On other hand the major drawback of this type is high amount of NOx in emissions. The gas temperatures generated in diffusion flames are as high as 2000°C and acceptable material temperature limits are maintained by air bleeds.

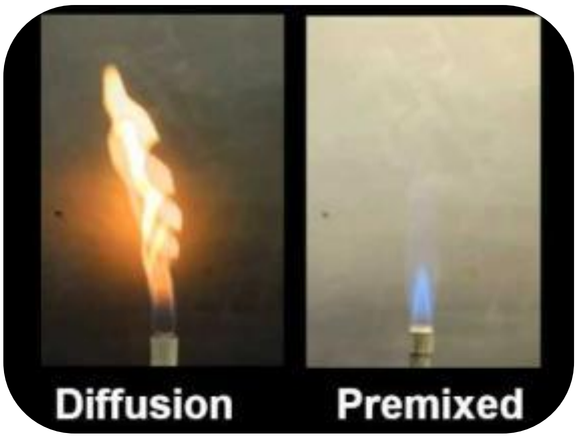


Fig5.1 Diffusion and Premixed flames [18]

The Dry-Low NOx or DLN combustion introduced the concept of partially pre-mixed flames. Fuel and air are mixed partially hence giving low equivalence ratio mixtures which gives lower temperatures and low NOx emissions. But the major drawback of this concept is flame stabilization and difficult to maintain suitable lower equivalence ratio over entire load profile of engine. Also flashback problem is a commonplace in partially pre-mixed flames.

6. Power Turbine

The next component arranged downstream in a gas turbine engine is power turbine. Turbine extracts the energy of hot expanding gases pushed out from combustion chamber and turns the compressor. Fig.6.1 shows a power turbine with its relative positioning in gas turbine assembly.

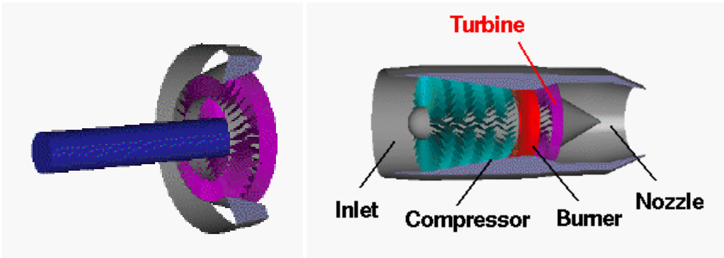


Fig6.1 Power turbine and its position in gas turbine assembly [5]

As shown in figure above, turbines are mounted on central shaft (blue color), which are connected to compressor on other end. The turbine, like compressors are made up of several rows of airfoil sets. Similarly, some of the sets are fixed which are known as stators which keep the flow from spiraling around by bringing it back parallel to the axis of rotation. Other sets which are connected to central shaft and rotate at high speeds are called as rotors.

Depending upon engine type, a multistage turbine can also be employed in modern day gas turbine. Turbofan and Turboprop engines use a separate shaft to power the fan and gearbox respectively. Such a configuration is known as two spool configuration. In some high performance machines there is even three spool configurations are applied, where an additional shaft is used to power different part of compressors. Fig. 6.2 below shows a two-spool turbofan configuration.

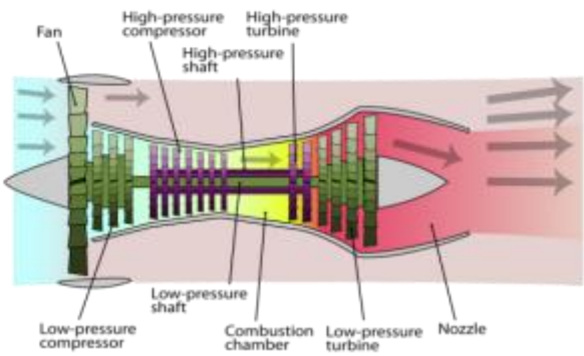


Fig6.2 Two spool turbofan engine [8]

6.1 Thermodynamics of power turbine

Part of the work generated by hot expanding gases through power turbine, is used to drive compressor. If the ratio of power needed to drive compressor to the power available in turbine is λ, then the energy balance for compressor turbine shaft is given as

$$W_c = \lambda W_t$$

Therefore in terms of temperature differences,

$$C_{pc}(T_{03} - T_{02}) = \lambda(1 + f)C_{ph}(T_{04} - T_{05})$$

$$\left(\frac{T_{05}}{T_{04}}\right) = 1 - \frac{(C_{pc}/C_{ph})T_{02}}{\lambda(1 + f)T_{04}}\left[\left(\frac{T_{03}}{T_{02}}\right) - 1\right]$$

and

$$\left\{\left(\frac{P_{05}}{P_{04}}\right) = 1 - \frac{(C_{pc}/C_{ph})T_{02}}{\lambda(1 + f)T_{04}}\left(1 + \frac{\gamma - 1}{2}M^2\right)\left[\left(\frac{P_{03}}{P_{02}}\right)^{(\gamma_c - 1)/\gamma_c} - 1\right]\right\}^{\gamma_h/(\gamma_h - 1)}$$

Due to mechanical losses and additional power required to drive accessories, adiabatic turbine efficiency η_t is employed in real case scenarios which in standard gas turbines range between $0.90 < \eta_t < 0.95$. Outlet pressure from power turbine is calculated by following relation,

$$\left\{\left(\frac{P_{05}}{P_{04}}\right) = 1 - \frac{(C_{pc}/C_{ph})T_{02}}{\lambda(1 + f)\eta_c\eta_t T_{04}}\left(1 + \frac{\gamma - 1}{2}M^2\right)\left[\left(\frac{P_{03}}{P_{02}}\right)^{(\gamma_c - 1)/\gamma_c} - 1\right]\right\}^{\gamma_h/(\gamma_h - 1)}$$

6.2 Design and operation of power turbine

Extraction of energy from a continuously expanding flow gives rise to a pressure drop across turbine. The resulting pressure gradient keeps the boundary flow attached to the turbine blades and hence making it less vulnerable to separation than on compressor. Therefore, pressure drop across a single turbine stage can be significantly greater than the pressure increase across a corresponding compressor stage. A turbine stage can be effectively used to drive multiple compressor stages.

It is also commonplace to find a thin metal band on the periphery of turbine blades which is helps in preventing flow leakage due to high pressure changes across blades. Turbine blades are exposed to much hostile conditions as compared compressor blades. Sitting downstream to combustion chamber, the temperature of hot gases may reach several thousand of degrees. High grade super alloys are being traditionally used in early turbine engines, but in modern day highly efficient gas turbine systems, even gas temperatures higher than that of melting point of conventional materials have been achieved by using advance active cooling concepts and new materials. Thermal Barrier Coatings (TBC) has also been used conventionally and undergoing a rigorous research to meet the future demands of high efficiency gas turbine engines.

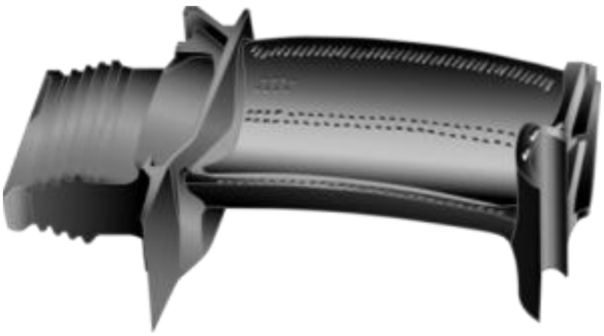


Fig.6.3 Power turbine blade with channels for active cooling [8]

6.3 Power turbine in micro gas turbines

Due to compact size, low cost, lower weight and simplicity of design a single stage turbine using conventional materials have been successfully used by most of the gas turbine manufacturers. Fig. 6.4 shows power turbine and centrifugal compressor mounted on central shaft of a micro gas turbine.

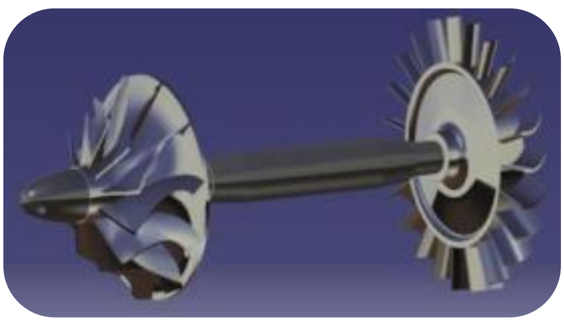


Fig.6.4 Shaft mounted power turbine and compressor in micro gas turbine [17]

7. Nozzle

All gas turbine engines have nozzles to produce thrust, release the exhaust gases back to free stream and to set mass flow rate through the engine. Nozzle sits next to power turbine downstream. A nozzle is relatively a simple device from design perspective, just as simple tube through which exhaust gases flow out, but it requires an involved thermo-fluid analysis to design a nozzle for a particular mission profile.

Depending on mission profile, nozzle comes in variety of shapes. Simple turbojet and turboshaft engine often use a fixed geometry convergent nozzle. Turbofan engines often use a co-annular nozzle. Core flow is exited through the central nozzle and fan flow is exited from annular nozzle. Mixing of the two flows provide some additional thrust enhancements and also tend to be quieter than simple convergent nozzle.

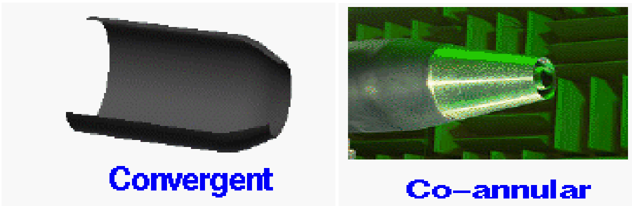


Fig.7.1 Convergent and co-annular type nozzles [5]

Afterburning turbomachines often employ a variable geometry convergent-divergent nozzle. In such nozzle, flow is first converges down to the minimum area or throat and next expands from throat towards exit. Flow is subsonic in convergent part but supersonic in divergent part. Due to additional mechanisms required to change nozzle geometry, such nozzles are heavier than the fixed geometry nozzles but provides efficient engine operation over wider range of airflow.

Fig7.2 Convergent divergent nozzle and rectangular nozzle of F-22 [5]

Recently, engineers have also been working on rectangular cross sectioned nozzles, which facilitates the easy vectoring or flow deflection which enhances the maneuverability of the aircraft. This factor is particularly important in fighter aircrafts.

7.1 Thermodynamic principle of nozzle

The exhaust velocity is obtained from the conservation of energy in nozzle. For both the cases, ideal or real, the nozzle must be checked whether it is choked or not which is based on calculating critical pressure from the relation,

$$\frac{P_{06}}{P_c} = \left(\frac{\gamma_h + 1}{2}\right)^{\gamma_h/(\gamma_h - 1)}$$

If the critical pressure P_c is greater or equal to ambient pressure, then nozzle is choked, therefore nozzle outlet temperature is given by,

$$\frac{T_{06}}{T_7} = \left(\frac{\gamma_h + 1}{2}\right)$$

exhaust velocity in this case is given by relation,

$$V_7 = \sqrt{\gamma_h R T_7}$$

In case where ambient pressure is greater than critical pressure, nozzle is unchoked and temperature of exhaust gases is determined from the relation,

$$\frac{T_{06}}{T_7} = \left(\frac{P_{06}}{P_a}\right)^{(\gamma_h - 1)/\gamma_h}$$

exhaust velocity in unchoked condition is calculated as,

$$V_7 = \sqrt{2C_p(T_{06} - T_7)}$$

Expansion process in the nozzle is similar to that in turbine and influenced by skin friction, hence governed by adiabatic nozzle efficiency, η_n which varies between $0.95 < \eta_n < 0.98$ for standard gas turbines. In real case, critical pressure P_c is calculated by relation,

$$\frac{P_{06}}{P_c} = \frac{1}{\left[1 - \frac{\left({}^{1}\!/_{\eta_n}\right)(\gamma_h - 1)}{(\gamma_h + 1)}\right]^{\frac{\gamma_h}{(\gamma_h - 1)}}}$$

for unchoked nozzle,

$$V_7 = \sqrt{\frac{2\gamma_h \eta_n R T_{06}}{(\gamma_h - 1)}\left[1 - \left(\frac{P_a}{P_{06}}\right)^{(\gamma_h - 1)/\gamma_h}\right]}$$

if nozzle is choked, then outlet temperature is calculated from relation,

$$\frac{T_{06}}{T_7} = \left(\frac{\gamma_h + 1}{2}\right)$$

and jet speed is calculated as,

$$V_7 = \sqrt{\gamma_h R T_7}$$

7.2 Nozzle in micro gas turbine

In micro gas turbine, due to simplicity in design, compact size, low cost and no special requirement on maneuverability or afterburning, simple fixed geometry convergent nozzle is sufficiently employed.

8. Examination of Key Advanced Combustion Chamber Geometries

The combustion chamber and combustion dynamics associated with it are the prime factors influencing the performance of any internal combustion engine. The chamber has to be designed so that a stable and continuous combustion process is ensured over the full operational profile of engine. Generally, combustion chamber is divided into three zones as shown in fig.5.3 before, primary zone, secondary or intermediate zone and tertiary or dilution zone. Major process of combustion takes place in primary zone; secondary or intermediate zone dilutes the combustion products and helps in carrying away soot, whereas in tertiary zone bleed air is mixed in hot combustion gases to reduce the temperature within the allowable limits of turbine material.

Dynamics of combustion process occurring inside a combustion chamber is a key factor in determining the combustion efficiency and emissions from the engine. To meet future regulations for emissions, gas turbine manufacturers are continuously looking for combustion technologies which can offer stable combustion, satisfactory level of emissions and with increased durability. Over the past few years more rigorous scientific efforts have been undertaken in particular to address NOx emissions. In general, to reduce NOx emissions while maintaining high combustion efficiency is not an easy task and it is always a trade-off between amount of NOx and Carbon compounds in engine emissions.

Though there are many traditional methods to reduce NOx emissions well below the allowable limits like steam addition also known as wet combustion. Though wet combustion have been traditionally used in earlier days of gas turbine development, but there is always a limit upto which steam can be fed, which otherwise leads to increased amount of carbon compounds in emissions. Also, due to additional cost and weight incurred to produce a high quality steam suitable for modern day turbine engines, researchers are motivated to look alternative technology and methods which can be a more optimal solution.

The above challenges can successfully be met by modification in combustion chamber design of today. Such combustion chamber designs are presented here and analyzed further for their suitability in micro scale subjected to technical constraints.

8.1 Trapped Vortex Combustion (TVC)

Trapped Vortex Combustion is one of the most promising combustion chamber geometry concepts which have gained a widespread attention in research for addressing lower pollutant emissions and pressure drop reductions.

8.1.1 Principle of operation

Trapped Vortex Combustion is based on simple concept of trapping the vortex by changing the combustion chamber geometry to a back step sudden expansion design. The combustion then takes place in this stabilized vortex zone by mixing fuel and air. Trapped Vortex Combustion concept has been studied since 1990's; mainly for the liquid fuel applications in aircraft engines.Fig9.1 below shows a simple schematic of a Trapped Vortex Combustion concept.

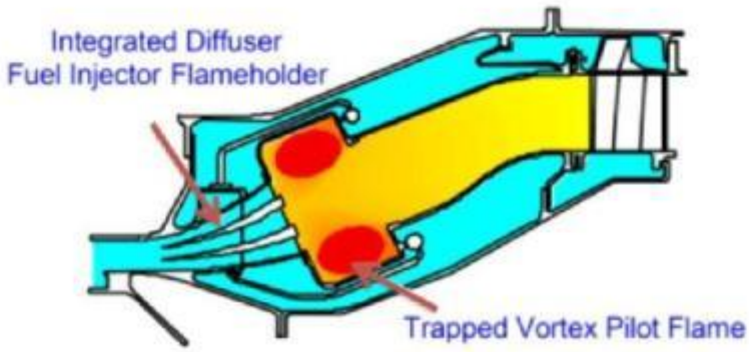


Fig8.1 Trapped Vortex Combustion (TVC) concept [2]

Trapped vortex technology offers several benefits as gas turbine burner:

- fuel flexibility- it is feasible to burn variety of fuels with medium and low calorific values including hydrogen rich gasified coal, biomass products and landfill gas
- very low NOx emissions are achievable without dilution or post combustion treatment
- improves the overall efficiency of gas turbine by decreasing the pressure drop though combustor
- extends the lean blow out limit with improved combustion and process stability

In TVC, trapped recirculation zone act as continuous source of ignition thus facilitating the efficient mixing of hot combustion products with incoming air-fuel mixture and leading to a stabilized flame zone. The vortex generated due to sudden expansion in gas flow is trapped in the cavity and gives a substantial reduction in pressure drop.

8.1.2 Trapped Vortex Combustion (TVC) at micro scale

As pointed out in previous chapters, relative ease and advantages of a centrifugal compressor at micro scale has made it omnipresent in micro gas turbine industry. Centrifugal compressor requires a spiral shaped casing or diffuser which changes the direction of out coming compressed radial air flow back into the axial direction before it enters into combustion chamber. Therefore, the simplicity in combustion chamber design facilitates lower costs of a micro turbine engines due to ease in manufacturing and assembly. Back step geometry used in TVC offers such simplicity and ease of manufacturing at micro scale

8.2 Rich Burn, Quick Mix, Lean Burn (RQL)

Lean Direct Injection (LDI) and Rich Burn/Quick Mix/Lean Burn (RQL) are two prominent techniques for ensuring low NOx emissions in gas turbines. Though both these concepts avoid stochiometric mixture ratios in primary zone, the very basic principle of operation is entirely different. LDI operates in lean primary zone, thus requires effective management to ensure flame stabilization. RQL concept operates on rich primary zone with a transition to lean combustion followed by rapid mixing with secondary air downstream. Fig7.2 shows a simple schematic of RQL with equivalence ratio and temperature dependence of production rate of nitrogen oxides.

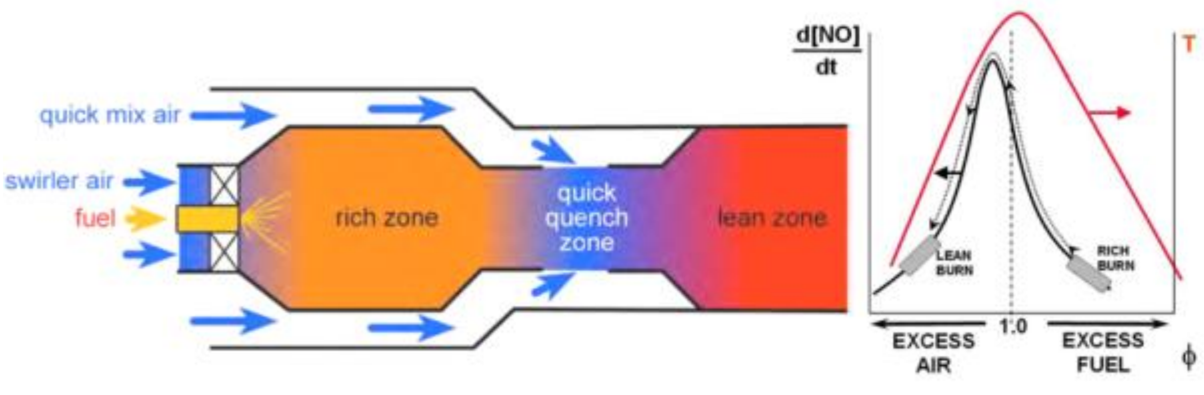


Fig8.2 Schematic of RQL and NO production rate dependence on equivalence ratio [2]

8.2.1 Principle of operation

Due to low temperatures and less amount of oxygen containing intermediate species, rich burn conditions in primary zone minimizes the synthesis of nitrogen oxides. The critical factors to consider in RQL are fast mixing rates and careful management of rich and lean equivalent ratios. The rapid mixing of secondary airflow to generate a lean burn zone also takes away the heat and thus reduces the effective temperature. Although, it must be ensured that the temperature of the combustion products in lean zone remain high enough to burn CO and UHC. Thus RQL system is highly sensitive to equivalence ratio to produce emissions well within the regulations. Typical equivalence ratio's of rich primary zone range between 1.2 -1.6 and for lean zone in range 0.5 and 0.7.

8.2.2 Rich Burn, Quick Mix, Lean Burn (RQL) at micro scale

As described above, to work effectively RQL requires rigorous efforts in tailoring the transition from rich primary zone to lean secondary zone. The cooling rate and thus the temperature of combustion products in lean zone are of critical importance, to strike the balance between NOx and CO/UHC emissions. According to author, at micro scale RQL poses two major drawbacks. First drawback is that this technology requires additional hardware and increases the complexity of combustion chamber and system as whole. Secondly, due to smaller dimensions of combustion chamber and high quenching rate the heat loss will be further accelerated thus posing a greater difficulty in maintaining temperature at required minimum level to avoid flame stability or even full blowout.

8.3 Continuous Staged Air Combustion (COSTAIR)

COSTAIR combustion uses continuously staged air and internal recirculation in combustion chamber to obtain a stabilized flame with low NOx and CO emissions. Staged air combustion is now been widely used in industrial systems but recently this system is gaining an attention for micro gas turbines as well.

8.3.1 Principle of operation

Principle of operation of staged air combustion is shown in Fig. 8.3 below. It consists of a coaxial tube, where air is fed through internal tube and equally distributed inside combustion chamber through an air distributor. Gas/fuel fed is from outer tube and around distributed air at two or more locations.

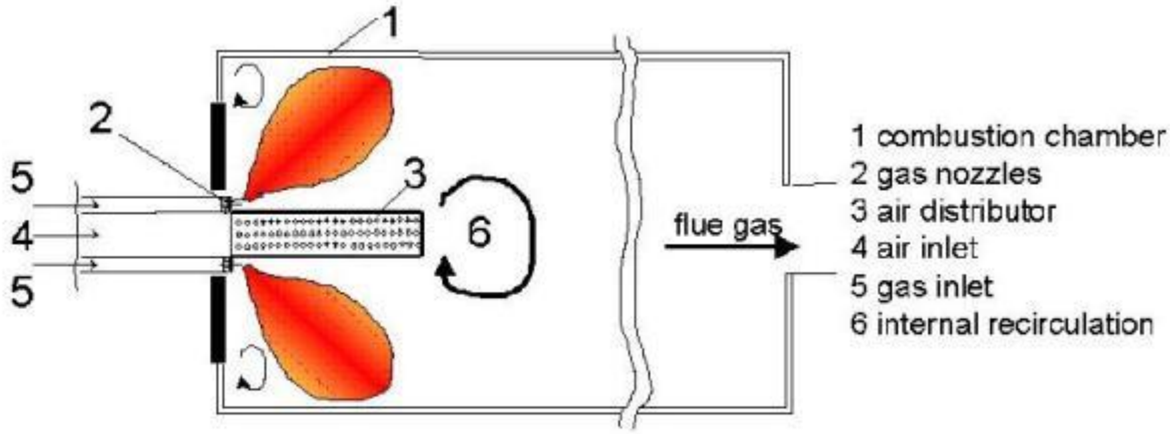


Fig8.3. Schematic of continuous staged air combustion [2]

Through this arrangement the heat is released more uniformly throughout the combustion chamber which leads to reduction in NOx formation notably. Injection of secondary air directly into combustion chamber results in internal recirculation of flue gases which enhances the NOx reduction effect. Flue gas recirculation reduces NOx production by reducing peak temperature in burner since diluting the charge with an inert gas decreases the adiabatic temperature.

The COSTAIR burner has advantages for working on diffusion mode or partially pre-mixed mode. It provides stable combustion process even with high air ratio as well as cold combustion chamber walls. A typical staged combustor has a lightly loaded primary zone, which provides the entire temperature rise needed to drive the engine at 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 high power settings, its main role is to act as a pilot source of heat for the main combustion zone, which is supplied with a 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 minimize NOx and smoke [7].

8.3.2 Types of staging

Staging type in combustion chambers has always been a choice between whether the staged combustion takes place in “series” or in “parallel”. The parallel configuration is known as radial staging and features the use of dual annular combustor as depicted in Fig. 8.4. One of these combustors is designed to operate lightly loaded and provide all temperature rise needed for start up, altitude re-light and engine idle conditions.

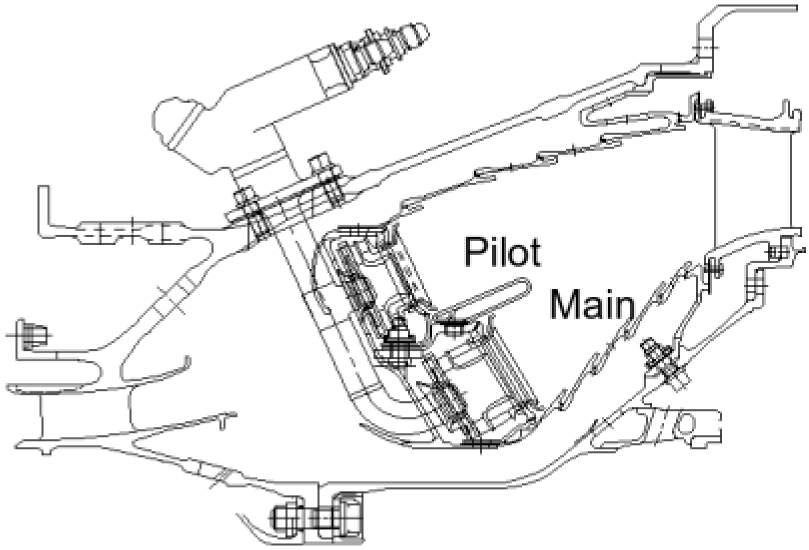


Fig.8.4 CFM 56-5B Dual annular combustor (radial staging) [9]

At idle, the equivalence ratio is so selected to minimize the emissions of CO and UHC. The other annular combustor is specifically designed to optimize the combustion process at high-power settings. It features a small, highly loaded combustion zone of short residence time and low equivalence ratio to minimize the formation of NOx and smoke. The DAC combustor shown in figure 8.4 is designed by GE and currently in service on Airbus A320 and A321. It achieved around 35% reduction in CO and UHC and 45% reduction in NOx as compared to corresponding single annular combustor.

In series or axial staging, a portion of fuel is injected into a conventional primary zone. Additional fuel is usually premixed with air in injected downstream into main combustion zone which operates at low equivalence ratios to minimize the formation of NOx and smoke. The primary zone is used on engine startup and generates the temperature rise needed to raise the rotational speed up to engine idle conditions. Fig8.5 below shows an axially staged combustor.

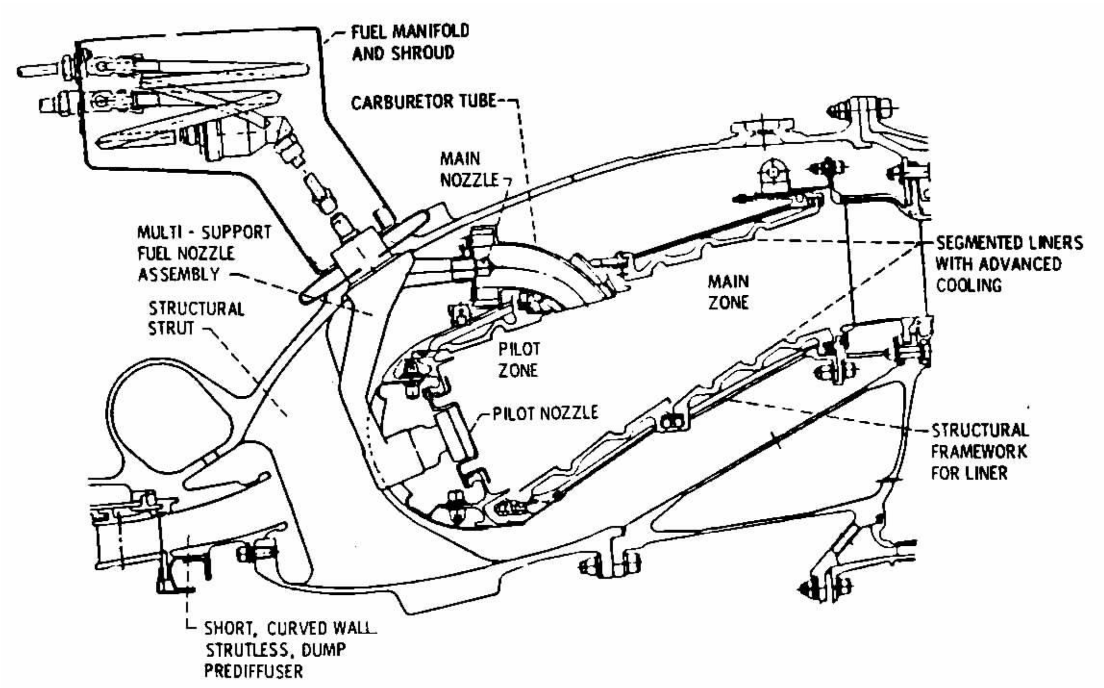


Fig 8.5 Pratt and Whitney axially stage combustor [16]

At higher power settings, fuel is supplied to the secondary combustion zone and as engine power rises towards its maximum value, the function of the primary zone becomes increasingly one of the heat sources for initiation of rapid combustion of fuel supplied to second stage.

8.3.3 Continuous Staged Air Combustion at micro scale

Major drawback of continuous staged air combustion concept is that enhanced combustion performance is achieved at the expense of increased design complexity and fuel injection hardware. Both staging concepts require highly specialized fuel injection systems to operate optimally. Therefore, the very design of COSTAIR concept presents challenges in meeting requirement of micro turbine technology primarily design and manufacturing simplicity, low weight and cost.

Taking particular staging designs into consideration for a micro gas turbine; radial staging suffers from basic problem of achieving the desired performance goals at intermediate power settings where both zones are operating well away from their optimum design points. This problem may tend to further intensify with decreasing dimensions at micro scale.

In axial staging the in-line arrangement of stages tends to create additional length which results in higher area for cooling. In micro scale, this problem may not pose a major challenge since heat loss rate is anyway higher due to smaller dimensions. But, increased complexity of fuel injector which requires separate arms for each combustion chamber and separate penetrations in combustor casings can present certain challenges at micro scale.

8.4 Moderate and Intense Low Oxygen Dilution combustion (MILD)

Moderate and Intense Low Oxygen Dilution or MILD combustion is one of the promising new technologies for clean and efficient combustion. It is clean and efficient relative to traditional combustion technologies due to uniform temperature distribution thus producing low NOx and CO/UHC emissions.

One of the very well known methods for improving combustion efficiency is to preheat the reactants by using hot flue gases. But this preheating leads to increase in NOx emissions due to increase in combustion temperature. Colorless distribution combustion is a technology which works on the middle ground between these two conflicting issues and produces low emissions with improved combustion efficiency.

This technology is also termed as Flameless Oxidation (FLOX), High Temperature Air Combustion (HiTAC) and Colorless Distributed Combustion (CDC) by different researchers. Though High Temperature Air Combustion (HiTAC) or FLOX is mainly studied for furnace applications, Gupta et. al. [8] have extensively worked on application for gas turbines and termed it as CDC.

8.4.1 Principle of operation

The successful application of MILD combustion requires fast mixing of exhaust gas recirculation, fresh air and fuel in combustion chamber before the combustion process begins. The recirculation of exhaust gases into fresh mixture of air and fuel raises its temperature over its auto-ignition temperature and thereby causing simultaneous ignition at different locations. Fig.8.6 below shows various combustion zones and their dependence on temperature, recirculation ratio and dilution of mixture.

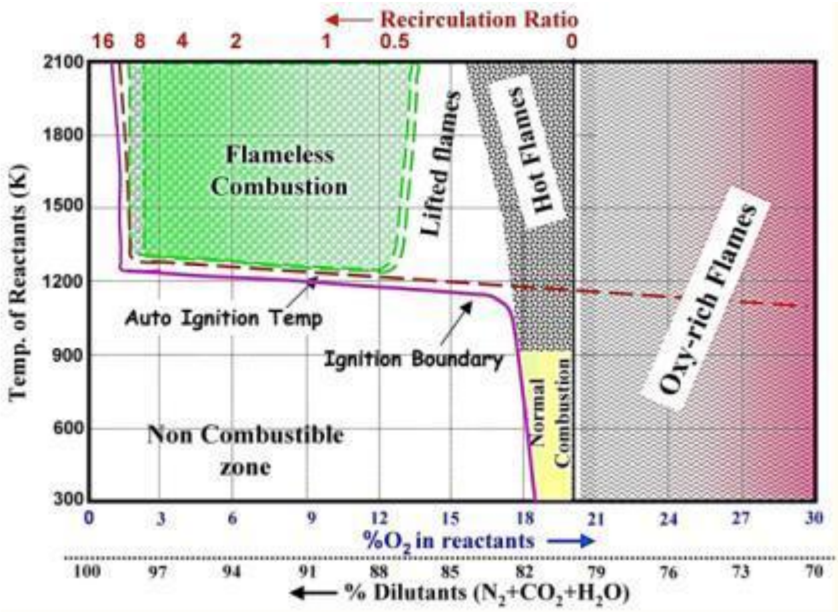


Fig.8.6 Different combustion zones [2]

Due to avoidance of small reaction zones and locally high peak temperatures, homogeneous combustion results in very low NOx emissions. Due to increased thermal intensities and missing heat transfer in gas turbine application, the conditions for MILD combustion takes place in higher exhaust gas recirculation temperatures. The higher temperatures result in increased reactivity and for MILD combustion to occur; faster mixing has to be achieved.

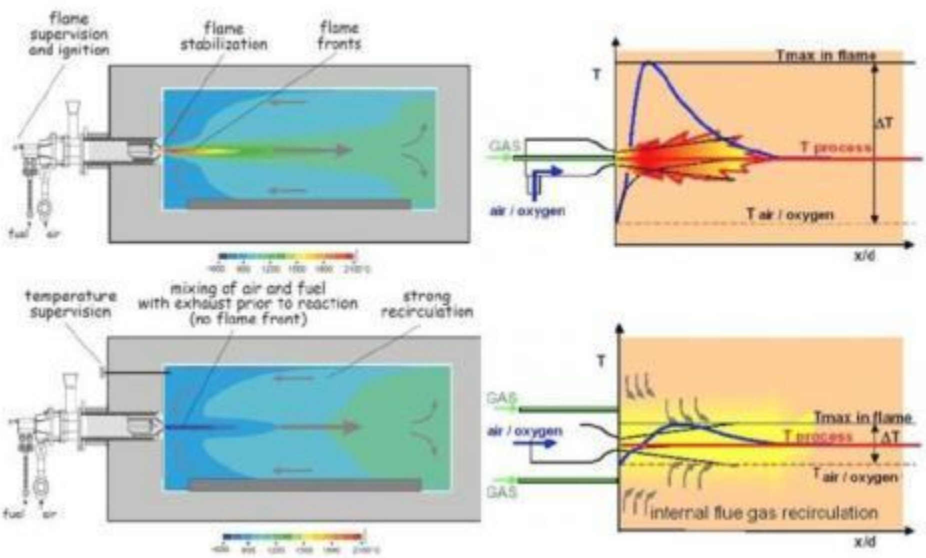


Fig.8.7 Schematic of MILD combustion [15]

8.4.2 Moderate and Intense Low Oxygen Dilution (MILD) combustion at micro scale

As evident from Fig.8.6 above, a high recirculation, temperatures of the range above 1500K and relatively high dilution is required to achieve a stable combustion in flameless zone. Complying with cost factor, traditional materials are widely used in micro gas turbines. Advanced materials of construction that can sustain the required ranges of temperature for MILD combustion may not only increase cost of micro gas turbines by many times but may also pose manufacturing challenges at micro scale.

Temperature sensitivity of MILD combustion can also present a major challenge at micro scale to sustain a stable combustion process. High surface to volume ratio in micro combustion chamber leads to flame quenching due to high heat loss through chamber walls. Since combustion mixture is diluted in MILD combustion, temperature loss below a certain limit will lead to sudden blow off as seen in Fig.8.6. Additional hardware required for dilution will also add weight, cost, and size penalty in a micro gas turbine system.

9. Proposed schematic of a micro gas turbine engine with Trapped Vortex Combustion chamber.

Figure 9.1 shows proposed schematic of a micro gas turbine engine with trapped vortex combustion chamber. It is worth to be mentioned here, that to channelize the compressed air flow parallel to the longitudinal axis of engine, a special conical shaped duct is proposed which is integrated with swirler. Figure 9.2 shows this conical duct-swirler assembly in details.

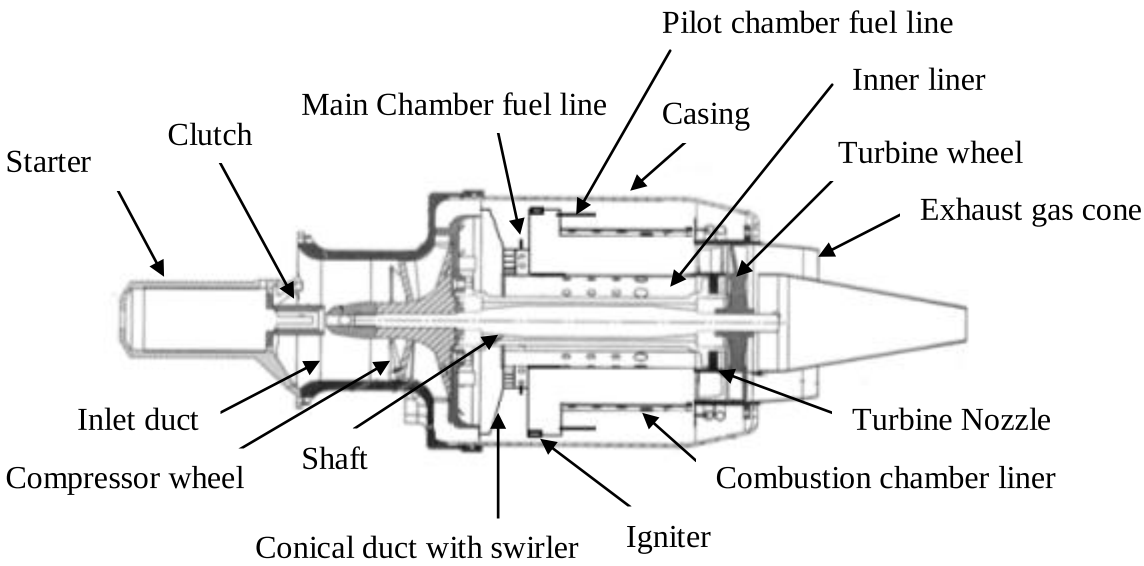


Fig 9.1 Proposed schematic of micro gas turbine with TVC chamber

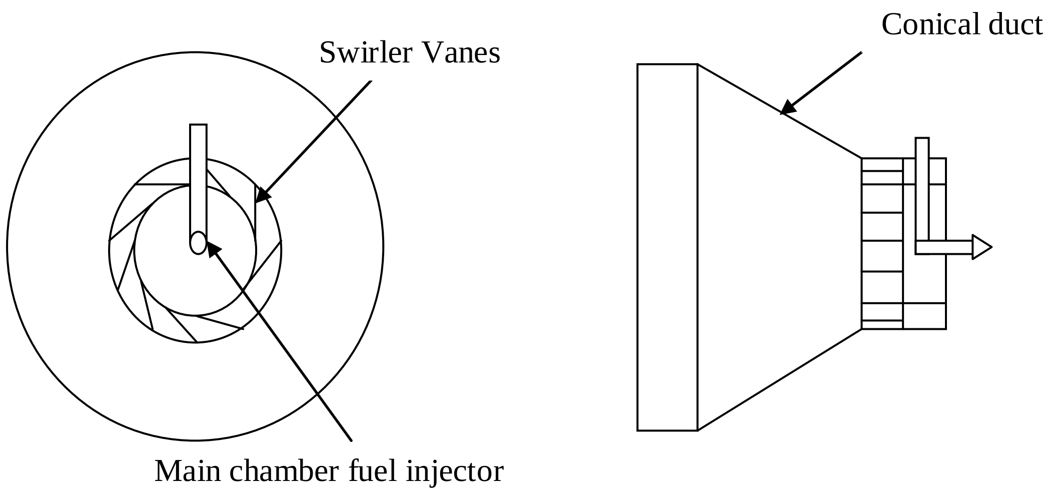


Fig. 9.2 Conical duct- swirler assembly

Referring to figure 9.1, the conical duct must be designed to capture 25-30% of the incoming compressed air and divert it to the longitudinal direction. Part of this captured compressed air enters the combustion chamber and expands through trapped vortex configuration where it is mixed with fuel and ignited in pilot combustion zone. Remaining part of captured compressed air passes through the inside liner and enters combustion chamber to dilute the combustion mixture.

70-75% of the incoming compressed air is bypassed and carries away the heat from pilot zone and further used to dilute the combustion mixture as it enters the main combustion zone through combustion chamber liner. As shown in figure 9.3, hot by pass air also undergoes sudden expansion due to trapped vortex geometry, which may introduce turbulence and hence facilitates better dilution of combustion products in main chamber. Hot by pass air may also tend to increase the temperature of pilot chamber inlet fuel as fuel lines are placed in its flow path.

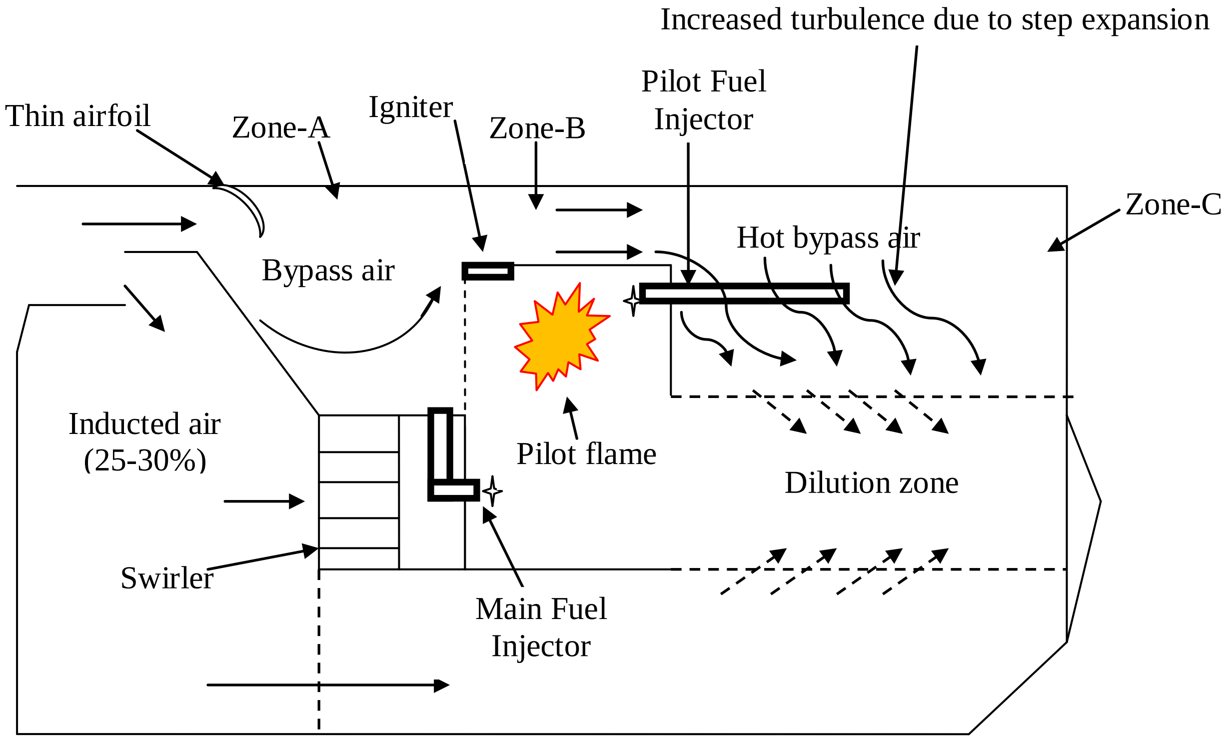


Fig. 9.3 Schematic showing compressed air distribution inside proposed configuration

As shown in figure 9.3, inducted air passes through a swirler where fuel is added to the stream before it enters combustion chamber. Swirler tends to further facilitate a better mixture and hence improved combustion process.

Effective heat transfer and appropriate cooling plays a critical role in combustion performance of a micro gas turbine engine. Referring to Fig. 9.3 above, ideally a vortex flow field in zone A, a parallel flow in zone B and a turbulent flow in zone C will facilitate the most effective heat transfer and cooling of combustion chamber.

As mentioned previously, the turbulence requirement of zone C, can be met to a greater extent due to sudden expansion of hot bypass air at the interface of zone B and C. To introduce vortex in zone A, it can be suggested to use an aerofoil shaped peripheral liner at the mouth of zone A (Fig. 9.3), which can direct the bypass air flow deeper into zone A and may produce a weak vortex in this zone. Introducing micro pores into the inner vertical edge of back step geometry (towards zone A), may also facilitate better cooling.

Transforming a weak vortex flow of zone A into a parallel flow in zone B will be presumably challenging to achieve. Optimization of conical duct-swirler assembly and outer edge of combustion chamber step may provide suitable conditions for effective cooling of combustion chamber. Fig. 9.4 below shows proposed elements and geometric features for optimization.

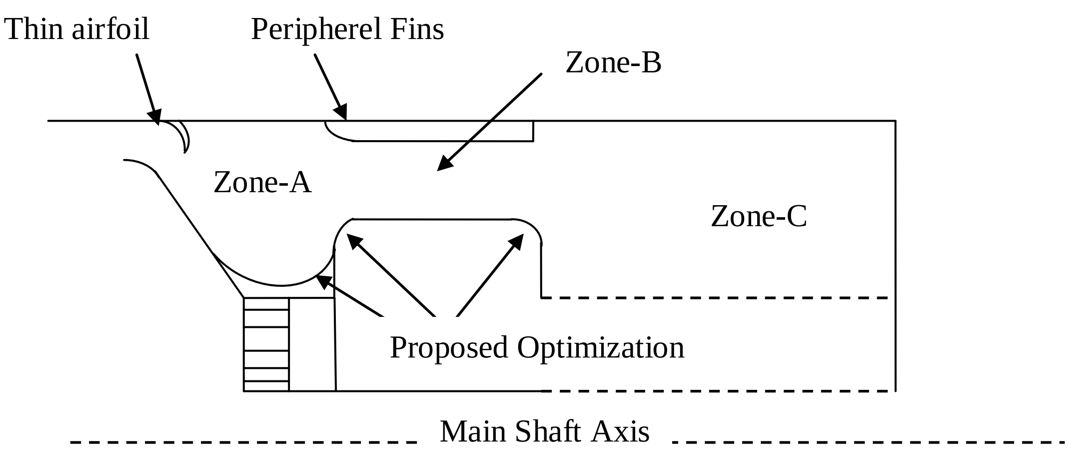


Fig 9.4 Proposed elements and geometric features to be optimized

As shown in Fig. 9.4 above, peripheral fins attached with casing is also proposed to facilitate the smooth entry of cooling bypass air into zone B and direct the flow parallel to the axis. Also, increased surface area due to linear fins will support better cooling.

9.1 CFD analysis of proposed combustion chamber configuration

Fluid flow process inside proposed combustion chamber configuration using Trapped Vortex Combustion concept is dealt in preliminary depth by Computational Fluid Dynamics (CFD). For the purpose of analysis self author has developed a personal CFD tool in Python programming language. A humble introduction to CFD concepts and tool itself is presented in next chapter. Simplified geometry considered for the analysis, applied boundary and initial conditions, study scheme and CFD results are presented in subsequent sections of this chapter.

9.1.1 CFD Tool

With the analysis of key advanced combustion concepts and their feasibility for a micro gas turbine presented in last chapter, a computational study for the most suitable concept has been conducted within this research work. Due to its simplicity in design, manufacturing and ease of application, author finds Trapped Vortex Combustion or TVC as the most feasible advanced combustion concept to apply for enhanced performance of a micro turbine engine.

To conduct a preliminary study on Trapped Vortex Combustion geometry, author has used a self written basic Computational Fluid Dynamics tool scripted in object oriented programming language Python. An insight is presented in this chapter about the tool itself bringing out details on principle equations, geometry and mathematics involved. Process of tool validation is also discussed.

Before delving into the details, author acknowledges that this tool is built by applying learning experiences from open course free lessons by Prof. Lorena A. Barbara from Boston University, titled " CFD Python: 12 Steps to Navier-Stokes" [10]. The code used in this study is a further development of the code presented in the above mentioned course to make it fit for this research work. In particular, additional energy equations, trapped vortex geometry and corresponding boundary conditions are programmed by author himself, applying the knowledge gained through above mentioned online course.

9.1.2 Computational Fluid Dynamics (CFD): A General Concept

Computational Fluid Dynamics or CFD can be rightly defined as third pillar of modern day aspect of Fluid Dynamic research, besides classical theory and experimental dimensions. CFD is an approach of using numerical computation mathematics to solve the governing flow equations for any given fluid flow situation, hence the name computational fluid dynamics.

The physical aspects of any fluid flow are governed mainly by three fundamental principles which are:

- Law of conservation of mass
- Newton's second law of motion (F = ma)
- Law of conservation of energy

These fundamental principles in their mathematical form are usually defined by set of partial differential equations. Broadly speaking, computational fluid dynamics is a method of replacing these governing equations with numbers and advancing these numbers in space and time to obtain final numerical description of complete flow field in control volume of interest [11].

In theoretical fluid dynamics, governing equations are solved over a continuous region of interest to present a solution for any given fluid flow problem. But in CFD approach, these governing equations are solved for discrete points in time and space domain within control volume. Hence, discretization of fluid dynamic equations is an important concept in CFD. Over the time, many discretization methods have evolved which are widely used in modern day engineering scenarios. Fig.9.5 below presents the basic concept of CFD.

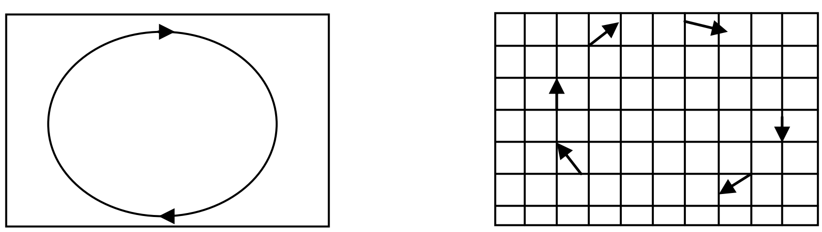


Fig. 9.5 Vortex in continuous flow field (left) and in discrete flow field (right)

9.1.3 Governing fluid dynamics: Navier Stokes Equation

Navier Stokes equations are one of the basic governing equations in fluid mechanics which were derived independently by G. G. Stokes in England and M. Navier in France in early 1800's. These equations describe the relation between velocity, temperature, pressure and density of a moving fluid. These equations are quite complex and extensions of Euler's equations with included viscosity effects of a fluid.

Navier-Stokes equations are set of coupled differential equations which can be solved using methods of calculus theoretically. But to solve these equations for a meaningful practical engineering problem is a cumbersome process. In past, engineers has made many approximations and simplifications to make these equations solvable for simple engineering problems but it was the recent advances in computer technology which could really enable the scientists and engineers to use these equations in practical situations.

Varieties of techniques like finite difference, finite volume, finite element etc. are currently being used by engineers to solve these equations using high speed computers. This area of study is what now popularly known as Computational Fluid Dynamics or CFD [5].

Broadly stating, Navier- Stokes equations are coupling of three basic laws of fluid mechanics (as mentioned in previous section) into one set of equations which are solved simultaneously to study mass, momentum and energy transfer during a fluid flow within an area of interest.

There are four independent variables, three space related x, y, z and one time related t. Six dependent variables include pressure p, density ρ , temperature T (contained in energy equations), velocity vector u, v and w for directions x, y, z respectively. Mathematical form of Navier-Stoke equations is described below.

Continuity: $\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$

X-momentum: $\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial \rho}{\partial x} + \frac{1}{Re}\left\{\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}\right\}$

Y-momentum: $\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial \rho}{\partial y} + \frac{1}{Re}\left\{\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z}\right\}$

Z-momentum: $\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial \rho}{\partial x}+\frac{1}{Re}\left\{\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right\}$

Energy: $\frac{\partial E_t}{\partial t}+\frac{\partial(uE_t)}{\partial x}+\frac{\partial(vE_t)}{\partial y}+\frac{\partial(wE_t)}{\partial z}=-\frac{\partial(\rho u)}{\partial x}-\frac{\partial(\rho v)}{\partial y}-\frac{\partial(\rho w)}{\partial z}-$

$\frac{1}{Re.Pr}\left\{\frac{\partial(q_x)}{\partial x}+\frac{\partial(q_y)}{\partial y}+\frac{\partial(q_z)}{\partial z}\right\}+\frac{1}{Re}\left\{\frac{\partial\left(u.\tau_{xx}+v.\tau_{xy}+w.\tau_{xz}\right)}{\partial x}+\frac{\partial(u.\tau_{xy}+v.\tau_{yy}+w.\tau_{yz})}{\partial y}+\right.$
$\left.\frac{\partial(u.\tau_{xz}+v.\tau_{yz}+w.\tau_{zz})}{\partial z}\right\}$

The terms on the left hand side of momentum equations are known as convection terms and that on the right side which are multiplied by inverse of Reynolds number “Re” are known as diffusion terms. Turbulence and Boundary layers are taken into account by diffusion terms. To solve a flow problem it is required to solve all these equations simultaneously thus Navier Stokes equations are said to be coupled equations. To specify all the terms of stress tensor, turbulence models are used in CFD [5].

9.1.4 Discretization technique: Finite difference method [11]

In contrast to analytical solution of partial differential equations, that gives variation of dependent variables in continuous manner within a domain, Numerical methods can lead to solutions only at discrete points or grid points within that domain. For example referring to Fig. 9.6 below, consider a section of discrete points in xy- plane.

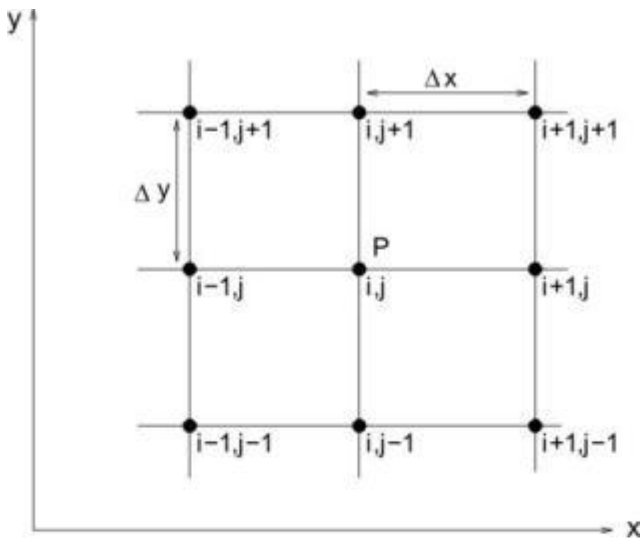


Fig. 9.6 Discrete grid points [12]

Assuming uniform distribution of points in x and y direction, the distance between two adjacent point in x direction Δx will be equal to distance in y direction Δy. In practice, Δx and Δy need not to be equal but makes calculations fast and convenient for simple flow field geometries.

As shown in figure 9.6, the grid points are identified by index *i* which runs in *x*-direction and index *j* which runs in *y*-direction. Therefore if (*i ,j*) is the index of point P in above grid, the point to its left will be denoted by index (*i-1,j*), point to its right will be denoted with index (*i+1, j*). Similarly, point directly above P will have index (*i, j+1*) and point directly below P will be denoted by index (*i, j-1*).

The method of finite differences which is widely used in CFD is also used here to discretize the partial differential equation sets of Navier-Stokes cluster and study the flow and temperature pattern in trapped vortex combustion geometry over discrete points or grids.

Finite difference representations of partial differential equations are based on Taylor's series. If $u_{i,j}$ denotes the *x*-component of velocity at point *i, j* , then the velocity $u_{i+1,j}$ at point *i+1,j* can be expressed in terms of Taylor series expanded around *i,j* as shown below.

$$u_{i+1,j} = u_{i,j} + \left(\frac{\partial u}{\partial x}\right)_{i,j} \Delta x + \left(\frac{\partial^2 y}{\partial x^2}\right)_{i,j} \frac{(\Delta x^2)}{2} + \left(\frac{\partial^3 y}{\partial x^3}\right)_{i,j} \frac{(\Delta x^3)}{6} + \ldots\ldots \qquad (9.1)$$

Equation 9.1 is the exact solution of $ui_{+1,j}$ if:

- number of terms are infinite and series converges
- and/or $\Delta x \rightarrow 0$

Higher order terms over third or fourth order are usually neglected because of the fact that most of the practical flow problems can easily be solved with reasonable amount of accuracy. Secondly, solving for higher order terms does not bring any significant addition in accuracy of solution but requires much more computer resources and time. Therefore, the above equation is truncated at second or third order and all the higher order terms are summed to give *truncation error* which is denoted by $O(\Delta x)$. Therefore truncated equation 9.1 for first order accuracy is denoted as:

$$u_{i+1,j} = u_{i,j} + \left(\frac{\partial u}{\partial x}\right)_{i,j} \Delta x + O(\Delta x) \qquad (9.2)$$

$$\frac{u_{i+1,j} - u_{i,j}}{\Delta x} + O(\Delta x) = \left(\frac{\partial u}{\partial x}\right)_{i,j} \tag{9.3}$$

Equation 9.3 is known as *first order forward difference*. Similarly, *x*-component of velocity for point *i-1, j* that is $u_{i-1,j}$ is given by expression:

$$u_{i-1,j} = u_{i,j} + \left(\frac{\partial u}{\partial x}\right)_{i,j} (-\Delta x) + \left(\frac{\partial^2 y}{\partial x^2}\right)_{i,j} \frac{(-\Delta x^2)}{2} + \left(\frac{\partial^3 y}{\partial x^3}\right)_{i,j} \frac{(-\Delta x^3)}{6} + \ldots\ldots \tag{9.4}$$

Solving eqn. 9.4 for $\left(\frac{\partial u}{\partial x}\right)_{i,j}$ and truncating it for first degree of accuracy we get,

$$\frac{u_{i,j} - u_{i-1,j}}{\Delta x} + O(\Delta x) = \left(\frac{\partial u}{\partial x}\right)_{i,j} \tag{9.5}$$

Equation 10.5 is known as *first order backward difference* expression for the derivative $\left(\frac{\partial u}{\partial x}\right)$ at grid points *i, j*. Second order derivative is obtained by *central difference* scheme as follows,

$$\frac{\partial^2 u}{\partial x^2} = \frac{u_{i+1,j} - 2u_{ij} + u_{i-1,j}}{(\Delta x)^2} \tag{9.6}$$

On parallel lines of finite difference method as shown in equations 9.1 to 9.6, full set of Navier Stokes equations are discretized in forward step time domain and backward step space domain to obtain the flow field in trapped vortex combustion chamber geometry.

9.1.5 Approximations and assumptions

Certain approximations and assumptions are made by author here to fit Navier Stokes equations for studying trapped vortex geometry. These approximations and assumptions are as follows:

- Flow in TVC geometry is studied in 2 dimensions (2D) [z- component = 0]
- Viscous effects in fluid flow are neglected [τ terms are ignored]
- Fluid is considered to be incompressible [ρ = constant]

Applying above three conditions and discretizing Navier-Stokes equation by using finite difference method in last section, final form used in CFD tool can be described as:

x-momentum

$$u^{n+1}{}_{i,j} = u^n{}_{i,j} - u^n{}_{i,j}.\frac{\Delta t}{\Delta x}\left(u^n{}_{i,j} - u^n{}_{i-1,j}\right) - v^n{}_{i,j}.\frac{\Delta t}{\Delta y}\left(u^n{}_{i,j} - u^n{}_{i,j-1}\right)$$
$$- \frac{\Delta t}{\rho.2.\Delta x}\left[\left(p^n{}_{i+1,j} - p^n{}_{i-1,j}\right) + \vartheta\left(\frac{\Delta t}{\Delta x^2}\left(u^n{}_{i+1,j} - 2u^n{}_{i,j} + u^n{}_{i-1,j}\right)\right)\right.$$
$$\left. + \frac{\Delta t}{\Delta y^2}\left(u^n{}_{i,j+1} - 2u^n{}_{i,j} + u^n{}_{i,j-1}\right)\right]$$

y-momentum

$$v^{n+1}{}_{i,j} = v^n{}_{i,j} - u^n{}_{i,j}.\frac{\Delta t}{\Delta x}\left(v^n{}_{i,j} - v^n{}_{i-1,j}\right) - v^n{}_{i,j}.\frac{\Delta t}{\Delta y}\left(v^n{}_{i,j} - v^n{}_{i,j-1}\right)$$
$$- \frac{\Delta t}{\rho.2.\Delta y}\left[\left(p^n{}_{i+1,j} - p^n{}_{i-1,j}\right) + \vartheta\left(\frac{\Delta t}{\Delta x^2}\left(v^n{}_{i+1,j} - 2v^n{}_{i,j} + v^n{}_{i-1,j}\right)\right)\right.$$
$$\left. + \frac{\Delta t}{\Delta y^2}\left(v^n{}_{i,j+1} - 2v^n{}_{i,j} + v^n{}_{i,j-1}\right)\right]$$

pressure-Poisson equation

$$p^n{}_{i,j} = \frac{\left(p^n{}_{i+1,j} + p^n{}_{i-1,j}\right).\Delta y^2 + \left(p^n{}_{i,j+1} + p^n{}_{i,j-1}\right).\Delta x^2}{2(\Delta x^2 + \Delta y^2)}$$
$$- \frac{\rho \Delta x^2 \Delta y^2}{2(\Delta x^2 + \Delta y^2)}\left[\frac{1}{\Delta t}\left(\frac{u^n{}_{i+1,j} - u^n{}_{i-1,j}}{2\Delta x} + \frac{v^n{}_{i+1,j} - v^n{}_{i-1,j}}{2\Delta y}\right)\right.$$
$$- \frac{u^n{}_{i+1,j} - u^n{}_{i-1,j}}{2\Delta x}.\frac{u^n{}_{i+1,j} - u^n{}_{i-1,j}}{2\Delta x}$$
$$- 2.\frac{u^n{}_{i,j+1} - u^n{}_{i,j-1}}{2\Delta y}.\frac{v^n{}_{i+1,j} - v^n{}_{i-1,j}}{2\Delta x}$$
$$\left. - \frac{v^n{}_{i,j+1} - v^n{}_{i,j-1}}{2\Delta y}.\frac{v^n{}_{i,j+1} - v^n{}_{i,j-1}}{2\Delta y}\right]$$

It is worth noting here that since in present case fluid is considered to be incompressible, there is no obvious way to couple the continuity equation with momentum equations. Therefore, pressure-Poisson ratio is used to satisfy the continuity while solving N-S equations simultaneously [10].

On similar lines, temperature distribution over grid is deduced from energy equation which in discrete form can be represented as:

temperature distribution

$$T^{n+1}{}_{i,j} = T^{n}{}_{i,j} - \frac{2u^{n}{}_{i,j}}{C.(1-2\gamma)} \cdot \left(\frac{u^{n+1}{}_{i,j} - u^{n}{}_{i,j}}{\Delta T}\right) - \frac{2v^{n}{}_{i,j}}{C.(1-2\gamma)} \cdot \left(\frac{v^{n+1}{}_{i,j} - v^{n}{}_{i,j}}{\Delta T}\right) +$$

$$\frac{2k\Delta t}{C.\rho.(1-2\gamma)} \left[\left(\frac{T^{n}{}_{i+1,j} - 2.T^{n}{}_{i,j} + T^{n}{}_{i-1,j}}{\Delta x^2}\right) + \left(\frac{T^{n}{}_{i,j+1} - 2.T^{n}{}_{i,j} + T^{n}{}_{i,j-1}}{\Delta y^2}\right)\right]$$

9.1.6 Code Validation: 2D Lid driven cavity problem [13] [14]

Lid driven cavity problem is one of the most used test cases used in industry and research to validate CFD codes. The problem geometry and boundary conditions are very simple which helps in easy and quick validation of any newly written code. Standard case of lid driven cavity test case is used here to validate the code written for this research work. Fig. 9.7 below shows test geometry used here.

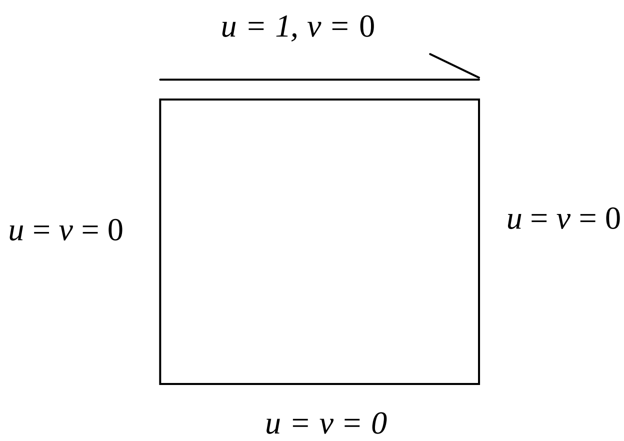
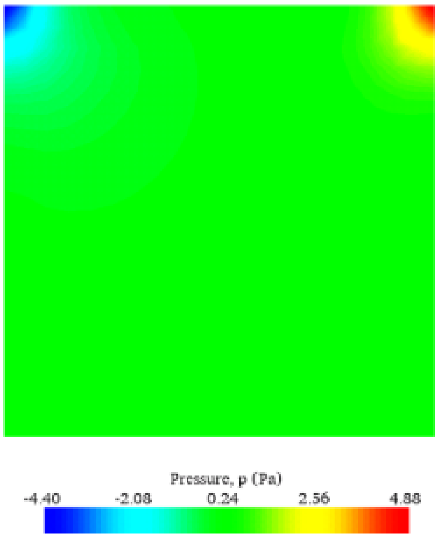


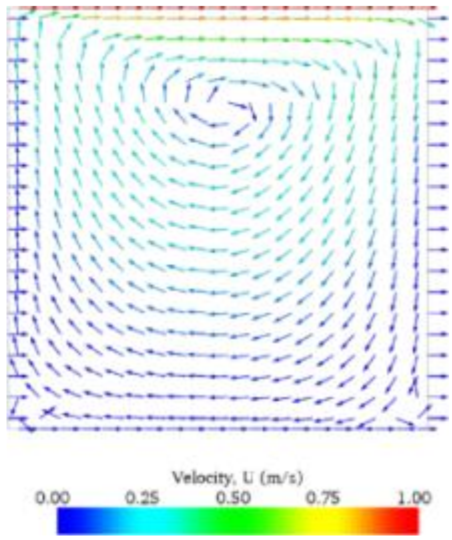
Fig.9.7 Standard lid driven test case geometry and boundary conditions

As shown in fig. 9.7 above, three sides of square cavity are stationary and the lid is moving in positive x-direction with a velocity of 1m/s. This problem is most suitable for code validation since sufficiently large amount of data and literature is available for this problem. Also, laminar solution for this problem is steady and boundary conditions are simple and compatible with most of the numerical approaches.

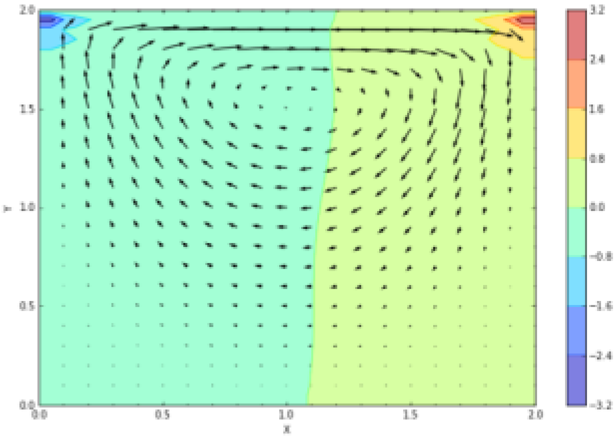
Lid driven cavity problem is solved on CFD code developed here and results are compared with that obtained in well known open source CFD code OpenFOAM. Result comparison is shown below in Fig.9.8



(a)



(b)



(c)

Fig.9.8 Code validation- Lid driven cavity a) Pressure profile in OpenFOAM b) Velocity profile in OpenFOAM c) Pressure and Velocity profile in CFD tool

From Fig.9.8 above, it can be deduced that CFD tool used here is working at quite similar level as an industry level CFD code for simple validation test case such as Lid Driven Cavity flow problem.

9.2 Simplified geometry of combustion chamber considered in CFD analysis

For CFD analysis of proposed combustion chamber configuration, simplified back step geometry is considered as shown in figure 9.9 below. It is important to mention here that CFD analysis is conducted in 2D, for incompressible flow and in absence of a swirler at combustion chamber inlet. The goal of study is to examine the flow behavior and bringing out an understanding on influence of structural changes in geometry of trapped vortex configuration upon combustion dynamics.

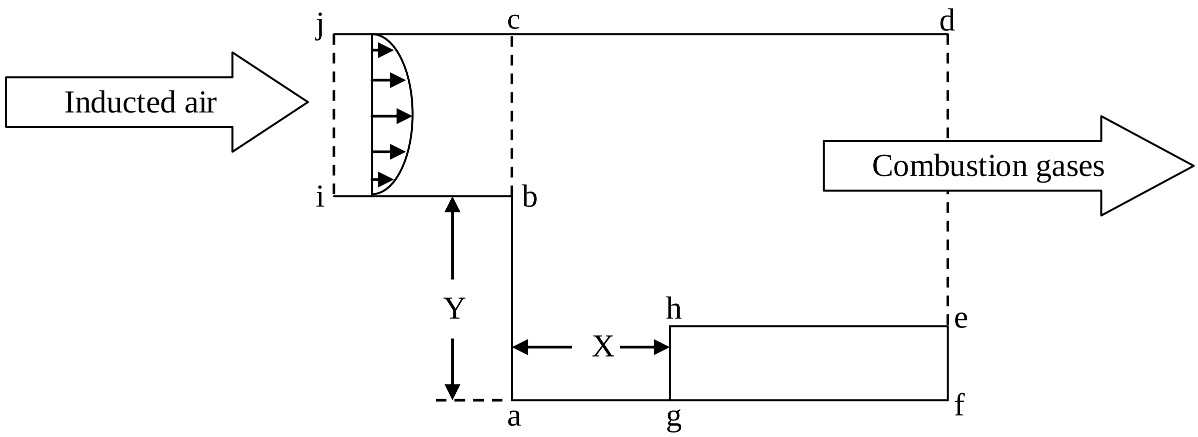


Fig. 9.9 Simplified back step geometry used for CFD analysis

Section “bijc” in figure 9.9 is inlet to the combustion chamber, where fully developed flow of inducted air is assumed. At the inlet edge “bc”, a uniform velocity $u = U_{mean}$ is assumed. Edge “jcd” represents the inner liner of combustion chamber. “bagh” is the sudden expansion step of geometry which is intended to generate and capture vortex.

In trapped vortex configuration, step ratio i.e. $\left(\frac{X}{Y}\right)$, plays a crucial role in generating a stable vortex and capturing it for considerable time to obtain an efficient fuel-air mixture. Presented CFD study aims to bring out relative importance of step ratio and its influence on flow field inside combustion chamber. Navier-Stoke equations are solved for section ‘abcdehg’, which is discretized into uniform square grids points of equal dimensions in both directions i.e. dx = dy.

9.3 Initial and Boundary conditions

Following initial and boundary conditions are applied,

Initial Conditions:

- Thermal Conductivity, k = constant
- C_p and C_v are constant
- incompressible flow, ρ = constant
- initial temperature T_i = 300 K

Boundary Conditions:

- at “bc” flow velocity in x-direction, u = U_{mean} , flow velocity in y-direction, v = 0
- at “ab” flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- at “ag flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- at “cd” flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- at “he” flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- at “gh” flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- in area “ghef ” flow velocity in x-direction, u = 0, flow velocity in y-direction, v = 0
- at “ag” , “he” and “cd”, pressure gradient $\frac{dp}{dy} = 0$
- at “ab” and “gh” , pressure gradient $\frac{dp}{dx} = 0$

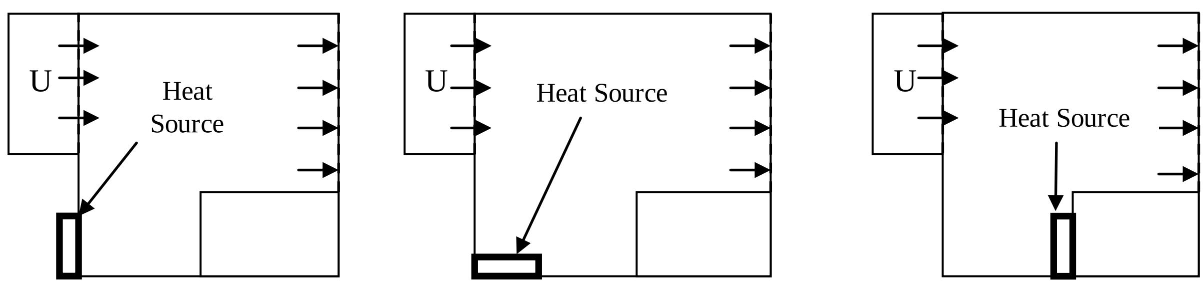
9.4 Study scheme

At first, Navier-Stokes equations are solved for a simplified back step geometry (Fig. 9.9) using a square grid with equal spacing in x and y direction. Heat losses due to radiation is neglected and constant temperature heat source at 1000K is considered. In current studies mainly two structural changes in chamber geometry are studied to investigate their influence on stable vortex generation. These two parameters are:

- step ratio $\left(\frac{X}{Y}\right)$
- heat source location

Flow field and temperature profile are examined from the solution of Navier-Stokes equation to bring out preliminary aero-thermal behavior of fluid flow inside a trapped vortex combustion chamber.

Configurations with step ratios varying from 0.5, 1.0, 1.5, and 2.0 (by varying step length- X) are studied here to bring out its influence on generation of a stable vortex and its efficient capture in cavity. Temperature variation of fluid flow inside combustion chamber is also studied for three different locations of constant temperature heat source to determine its influence on formation of potentially stabilized flame zones. These locations are described in figure 9.10 below. It is worth to be noted here that temperature profile analysis is presented using step ratio, which is examined as most suitable for generating stable vortex with its efficient capturing.



a) T1 b) T2 c) T3

Fig. 9.10 Heat source locations studied: T1, T2, and T3

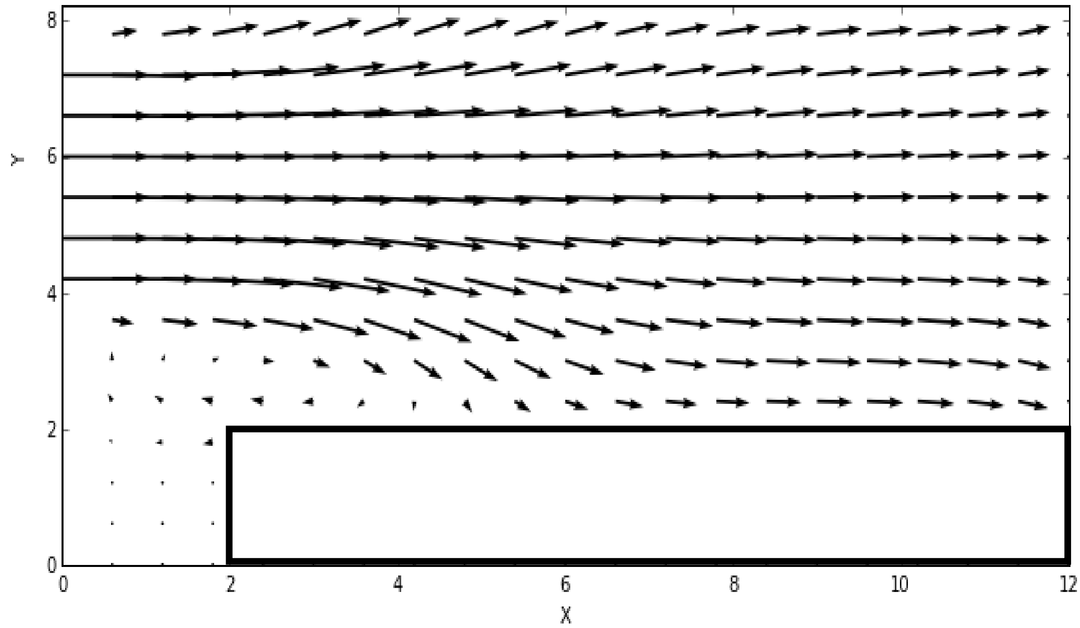
In addition to above mentioned two geometric parameters namely, step ratio and heat source location, role of inlet velocity "U", over structure of generated vortex is also studied. For this analysis four different indicative velocities in increasing order are provided at inlet and its influence over vortex formation is studied for step ratio corresponding to most suitable step geometry.

It is worth noting here that only indicative velocities are used in this analysis and not the actual velocities which may be encountered in practical situation for micro gas turbine engine. Due to simplified geometry, assumptions of incompressible and inviscid fluid flow, a converged solution cannot be obtained using the simple CFD tool used. Basic idea here is to bring out the influence of inlet velocity over vortex geometry for a given step ratio. Influence of source temperature is also examined for 1500 K and 2000K respectively.

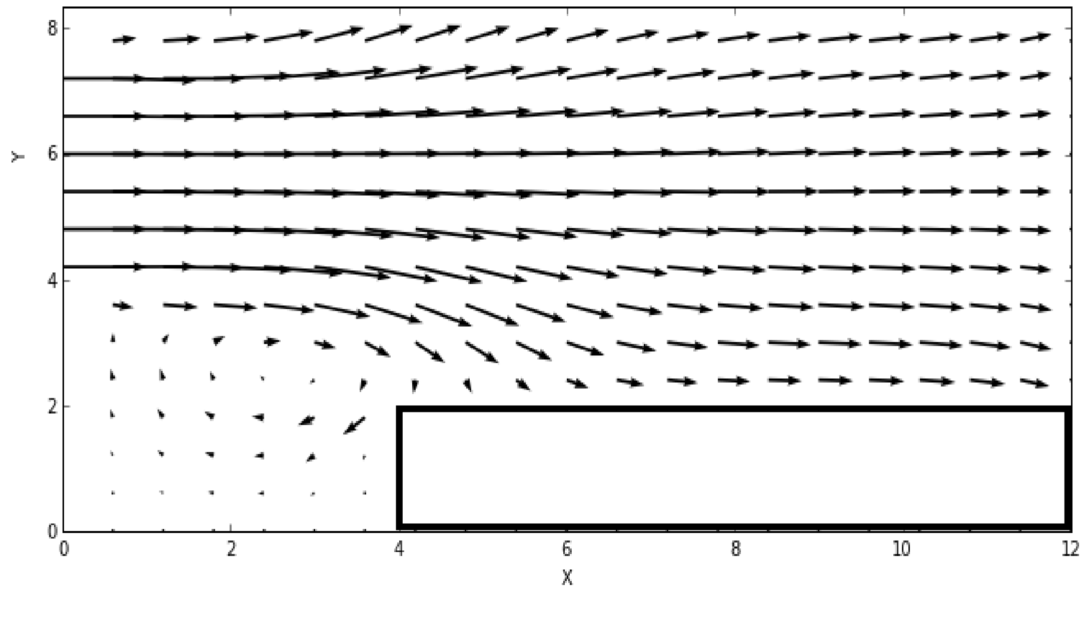
9.5 CFD analysis results

9.5.1 Flow field analysis for trapped vortex geometry with varying step ratio.

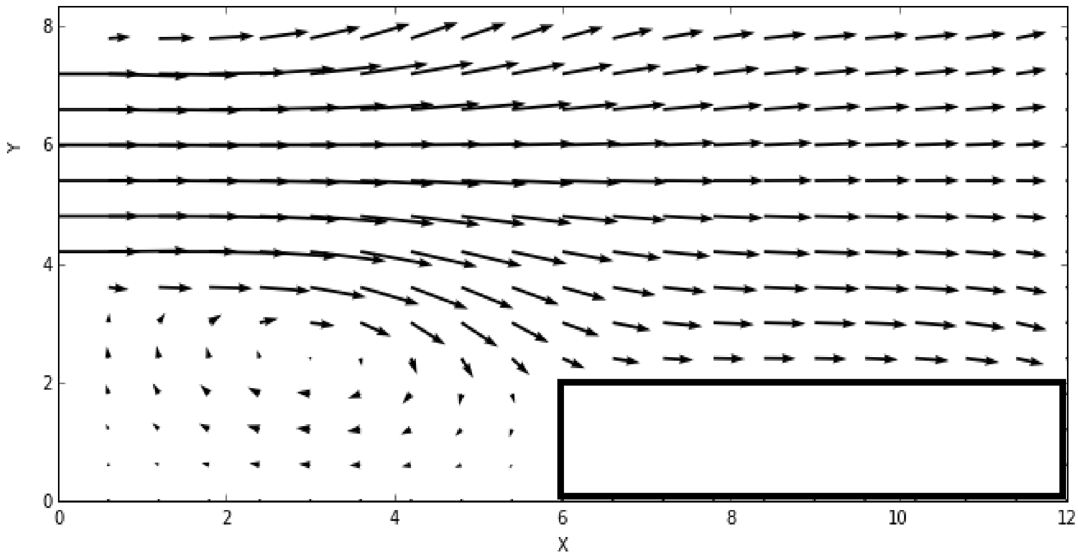
a) Step ratio, $\left(\frac{X}{Y}\right) = 0.5$



b) Step ratio, $\left(\frac{X}{Y}\right) = 1.0$



c) Step ratio, $\left(\frac{X}{Y}\right) = 1.5$



d) Step ratio, $\left(\frac{X}{Y}\right) = 2.0$

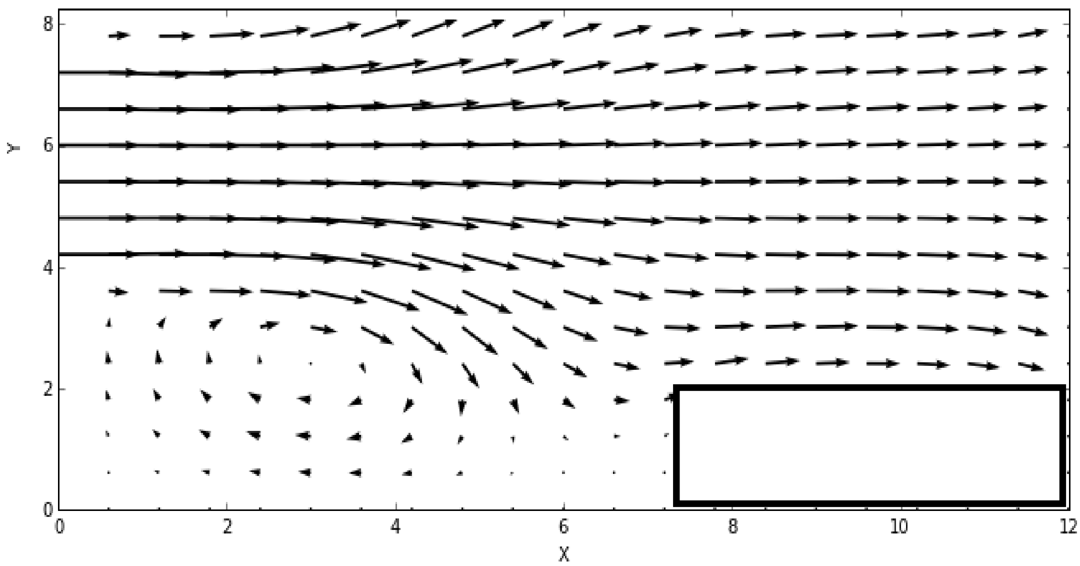


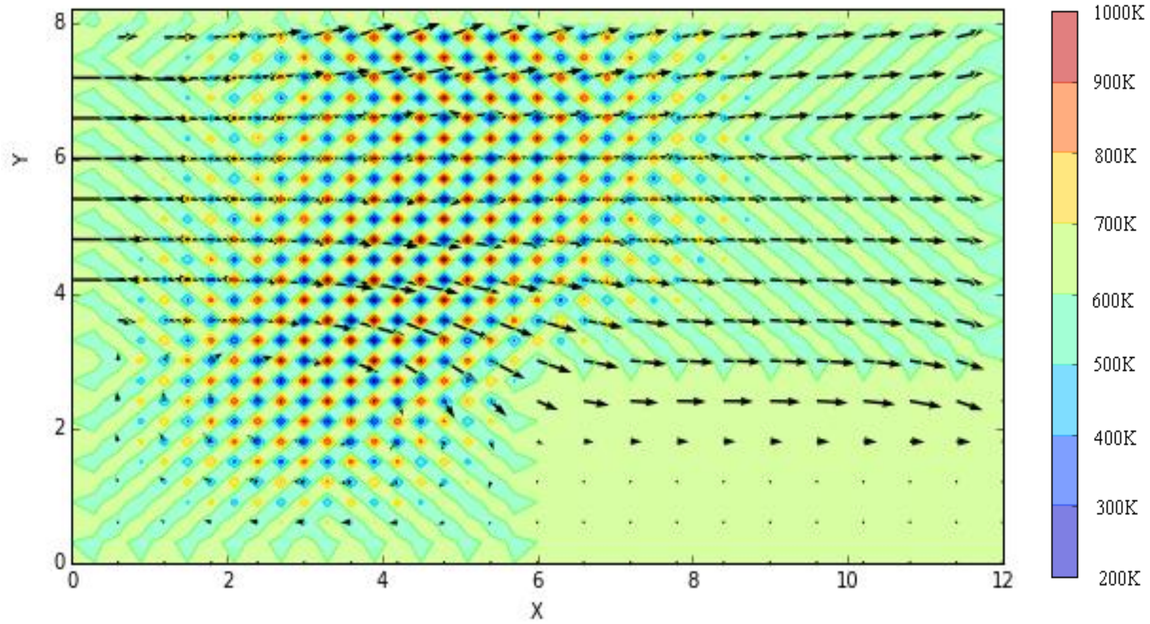
Fig. 9.11 Flow field for different step ratios

From above analysis, the influence of step ratio is evident in generating a stable vortex in pilot zone and capturing it for sufficient duration to allow efficient mixing of reacting species. With step ratio 0.5 (figure 9.11 (a)) and 1.0 (figure 9.11(b)), it can be observed that a smaller vortex is generated outside the step, therefore decreasing the probability of its capture inside cavity.

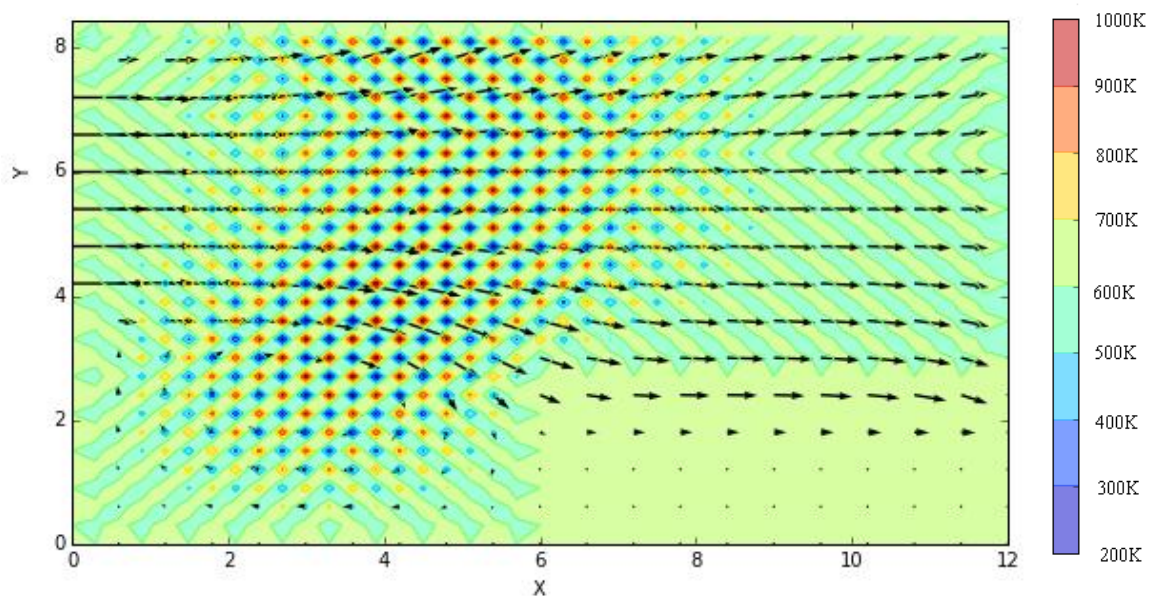
On other extreme, back step geometry with step ratio 2.0 (figure 9.11 (d)) generates a vortex with larger radius but this vortex may tend to escape from cavity since cavity length is much bigger than vortex major axis. Step ratio of 1.5 (figure 9.11(c)) gives the most appropriate conditions which tends to generate a vortex of sufficiently large radius, with capabilities to smoothly capture it inside cavity. In second part of this study, temperature profile for trapped vortex geometry is analyzed for step ratio 1.5.

9.5.2 Temperature profile analysis for trapped vortex geometry for heat source locations T1, T2 and T3 (Fig. 9.10)

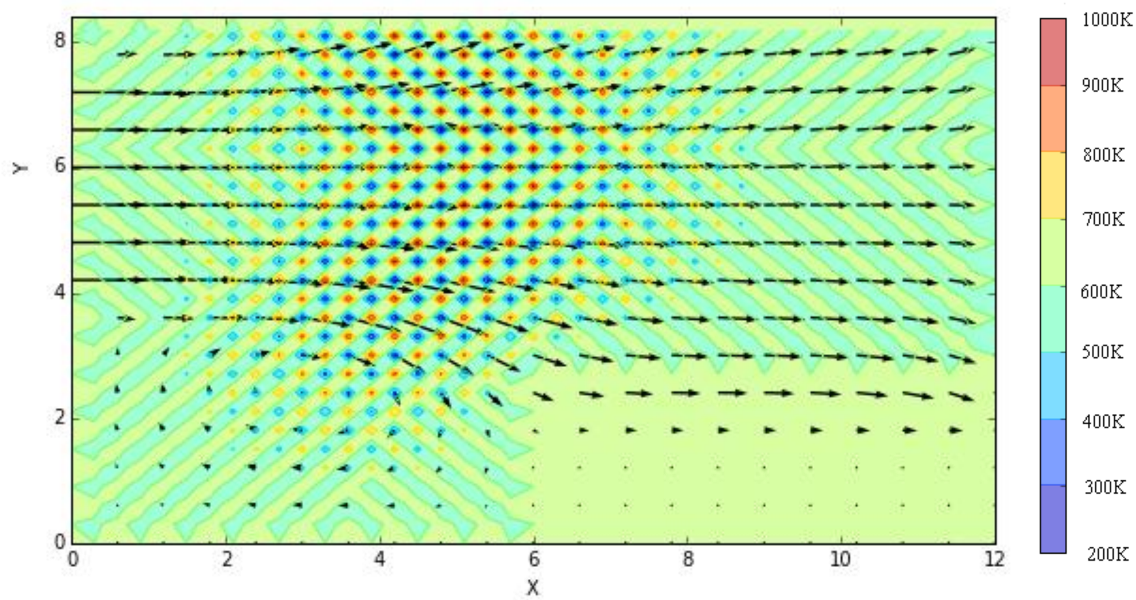
a) Heat source location T1



b) Heat source location T2



c) Heat source location T3



d) Heat source location T1+T2

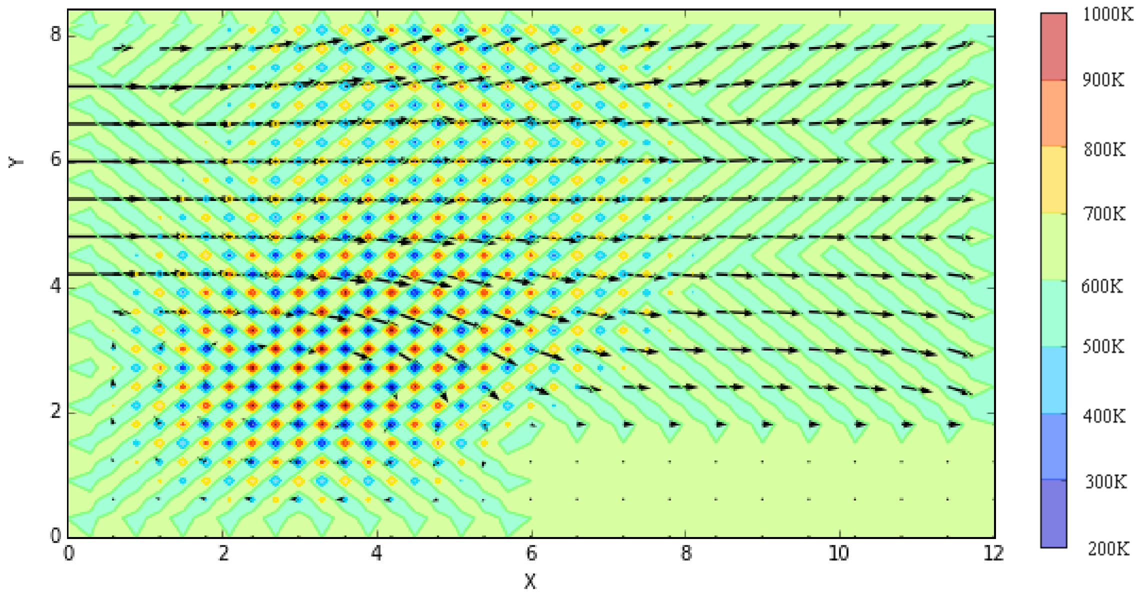


Fig. 9.12 Potential stable flame sites for different heat source locations

From temperature profile analysis for three different locations of heat source T1, T2 and T3 (fig. 9.10), relative importance of location at which heat source (igniter) is placed can be observed.

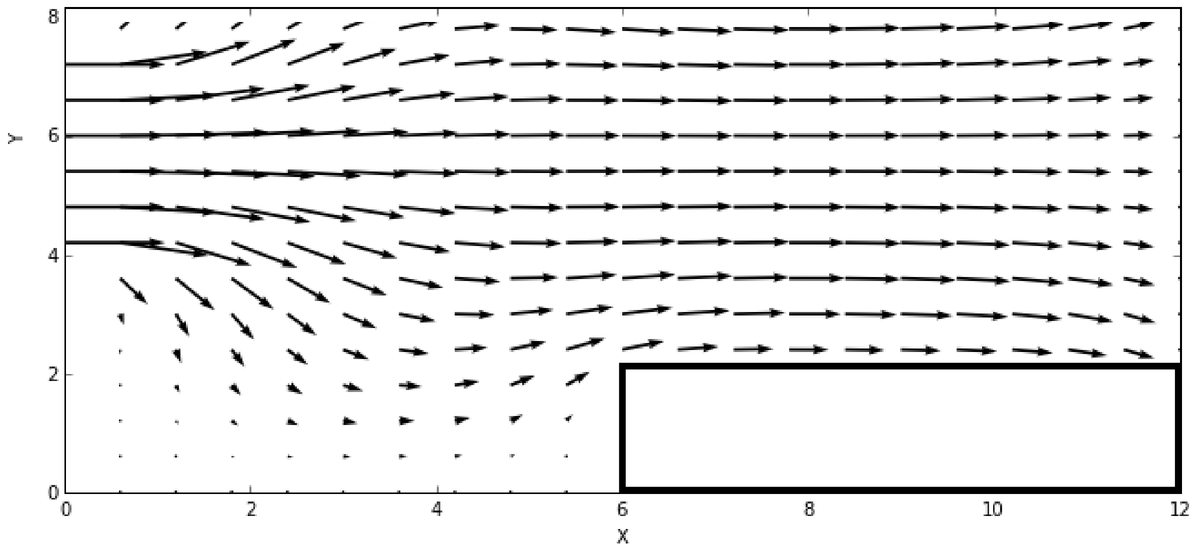
As seen in figure 9.12 (a)-(b), for heat source location T1 and T2, quite similar temperature profile is generated with potential flame sites spread in a direction from eye of the vortex in pilot zone to parallel flow in main combustion zone. In author's opinion, such a temperature profile emphasizes the advantage of trapped vortex combustion concept in micro gas turbine. Flame generated in a fuel rich pilot zone, spreads and supports the burning of lean mixture in main combustion zone.

Since potential stable flame sites are entirely situated almost outside the vortex zone in high velocity area, for heat source location T3 (figure 9.12 (c)), it may not allow for sustainability of a stable flame, on the contrary it may lead to complete blow off.

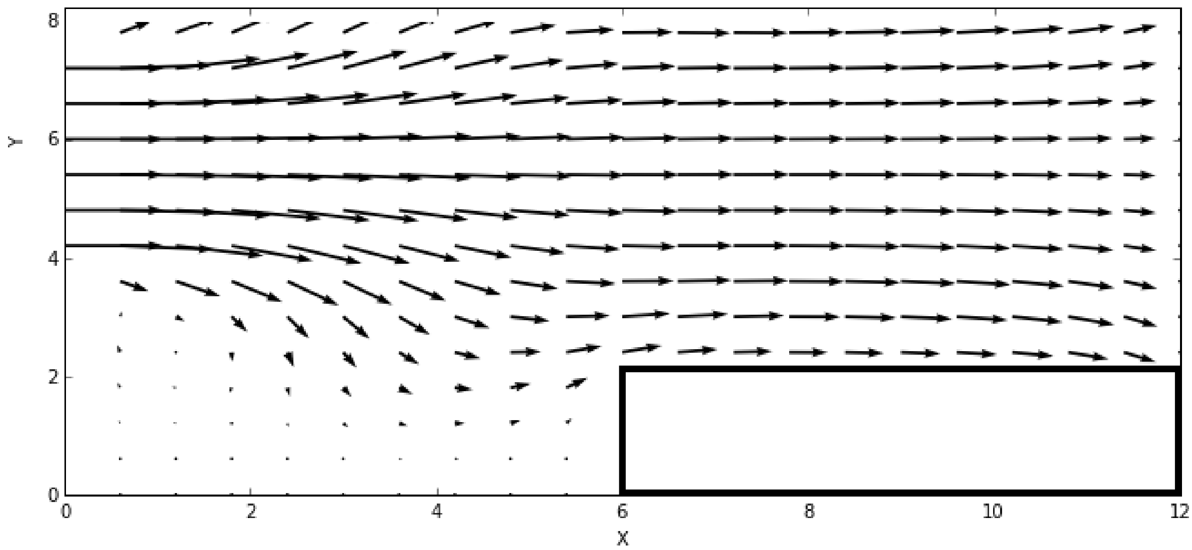
To take advantage of low velocity vortex trapped in cavity, author suggests to uniformly spreading the heat source at T1 and T2 location. It is quite evident from the analysis in figure 9.12 (d) for heat source location T1+T2 that this approach tends to effectively use the vortex zone.

9.5.3 Analysis of inlet velocity over vortex geometry

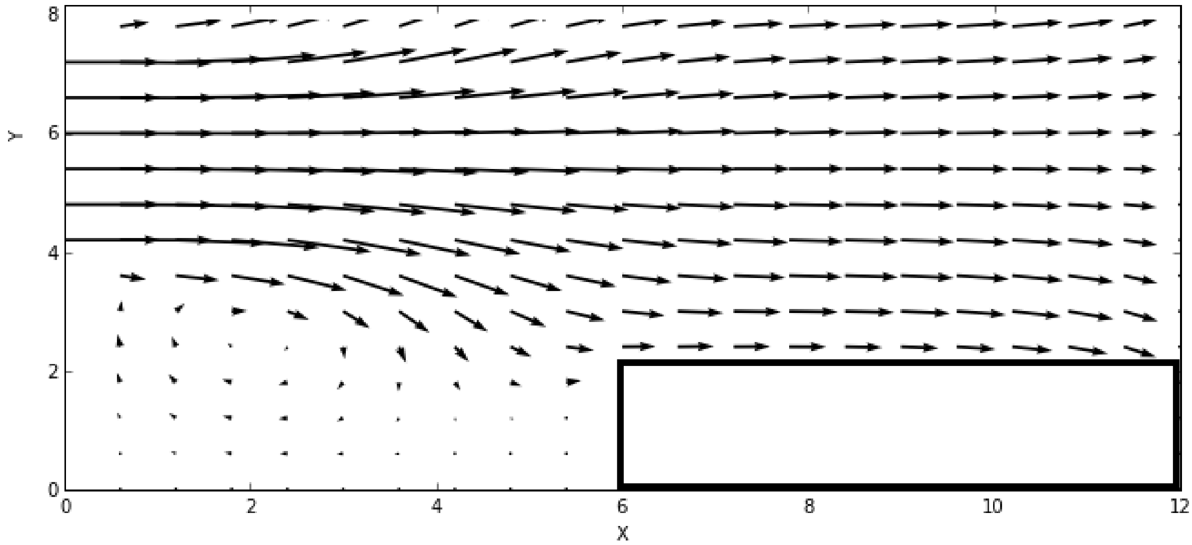
a) Indicative inlet velocity U = 1, step ratio = 1.5



b) Indicative inlet velocity U = 3, step ratio = 1.5



c) Indicative inlet velocity U = 5, step ratio = 1.5



d) Indicative inlet velocity U = 7, step ratio = 1.5

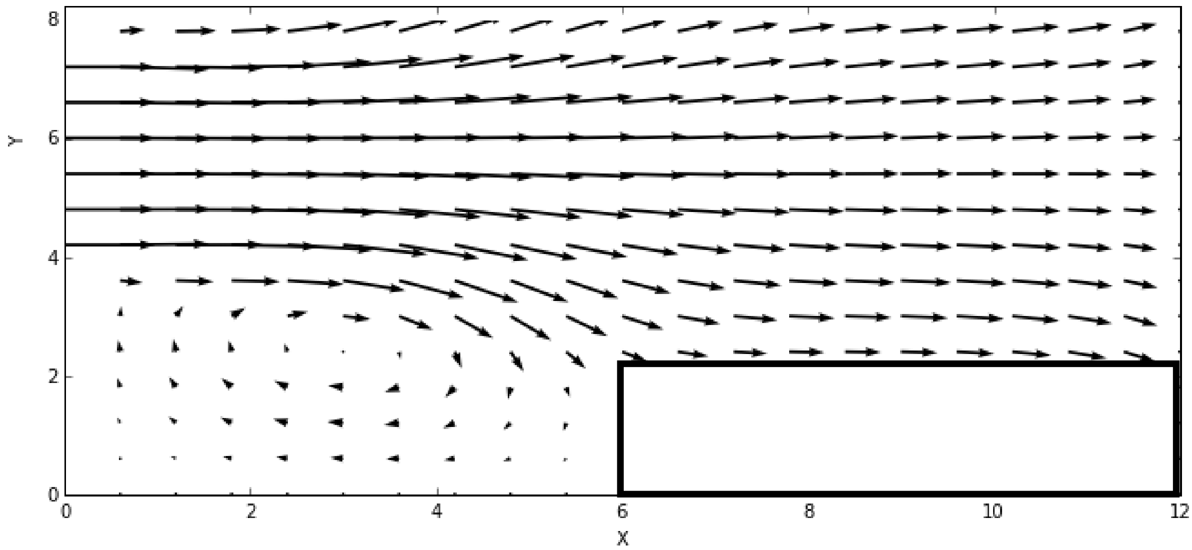
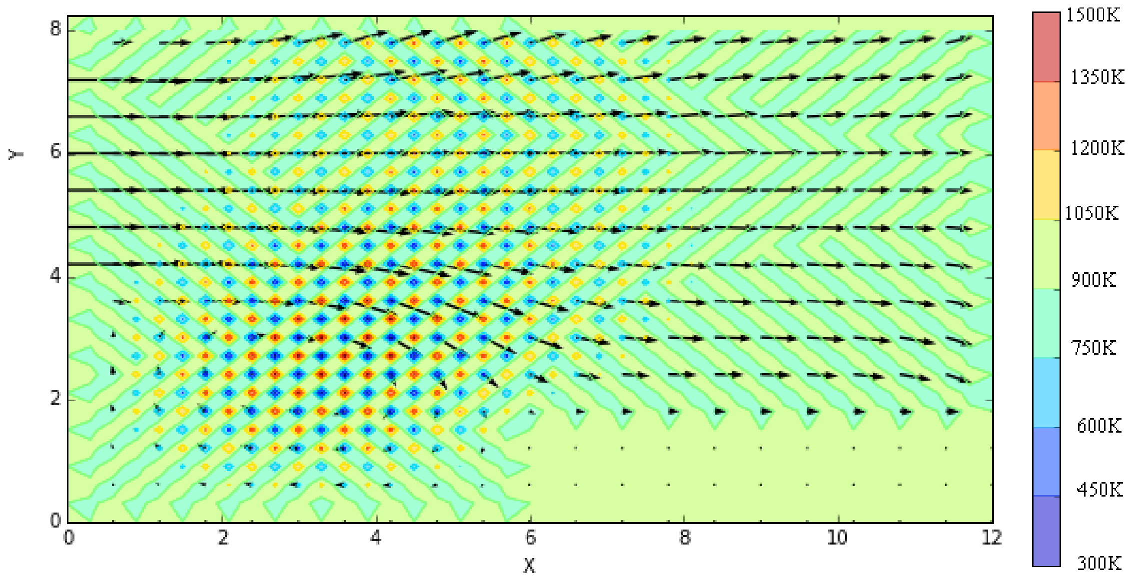


Fig. 9.13 Analysis of inlet velocity on vortex geometry for a given step ratio

Analysis shown in Fig. 9.13 above, throws light on critical significance of inlet velocity is generation of a stable vortex. Hence, for generating a useful vortex in TVC chamber micro gas turbine, a certain minimum velocity must be ensured. For the full operating range of micro gas turbine, trapped vortex combustion chamber must be designed to generate a stable vortex in pilot zone.

9.5.4 Analysis of source temperature on potential stable combustion temperature profile

a) Heat source temperature = 1500K, location = T1+T2, step ratio = 1.5



b) Heat source temperature = 2000K, location = T1+T2, step ratio = 1.5

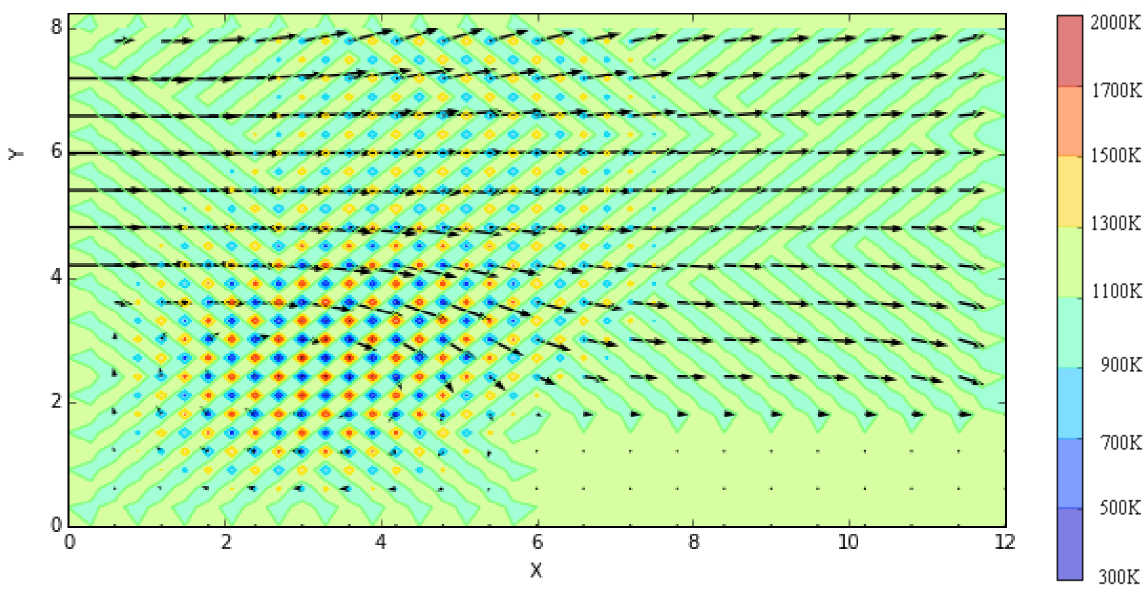


Fig. 9.14 Analysis of source temperature on potential stable combustion temperature profile

From the analysis presented above in Fig. 9.14, influence of heat source temperature can be clearly seen over the distribution of temperature field inside trapped vortex combustion chamber.

In practical scenario, heat source temperature depends upon various factors like fuel type, mass flow rate of fuel etc. Since mass flow rate is strongly determined by load on engine, therefore temperature inside trapped vortex combustion chamber will change over operating range of engine.

From Fig. 9.14, it can be observed that with increase in source temperature, the potential sites of combustion are concentrated more towards eye of trapped vortex. This is an important requirement to effectively utilize low velocity vortex zone to sustain a pilot flame. On other hand, increasing heat source temperature beyond certain limit is constrained by temperature limits of combustion chamber material.

Hence it can be concluded that to gain a meaningful advantage in performance of a micro gas turbine engine using trapped vortex combustion chamber, it is essential to maintain heat source temperature within the range which gives suitable conditions for sustaining a stable flame in pilot zone and main combustion zone, over entire operating range.

10. Conclusions and Propositions

In this thesis research work, a step by step analysis of influence geometrical elements on performance of each micro gas turbine element is presented. An effort has been made to bring out the understanding of bottlenecks and fundamental problems involved in designing gas turbine system elements at micro scale.

The basic question to achieve enhanced performance in terms of better fuel efficiency and possibly lower emissions in micro gas turbine is the central theme of this work which is addressed by inculcating possible changes in geometry of system elements. Conclusions drawn from this study and proposed improvement strategies are presented in this section.

10.1 Inlet

Inlets play an important role to ensure smooth, undisturbed and sufficient delivery of atmospheric air into the combustion chamber. In micro turbine engine inlet, two major design constraints can be listed. First, which is associated with technical limitation and simplicity is position of starter motor at inlet front which not only generates additional drag but also introduces large disturbances in inlet flow. Second limitation is associated with fundamental fluid dynamic behavior i.e. boundary layer separation of inlet flow due at micro dimensions of inlet. These two major constraints in addition to losses due to skin friction largely limit the efficiency of inlet in micro gas turbines.

10.1.1 Propositions

Geometrical design optimization: As a practical solution strategy for the above mentioned constraints in inlet design, geometrical design optimization can be a good method to improve inlet flow characteristics. With the availability of many open source and standard optimization codes, author proposes optimization studies of casing geometry of starter motor with mounting struts for the maximum expected inlet velocity of micro gas turbine.

One such optimization technique which author finds promising is Genetic Algorithms which are already finding widespread applications in design optimization problems in various engineering domains.

Peripheral Openings on inlet surface: Based on almost similar concept as variable lip inlet geometry in standard aircraft gas turbines, limited number of small openings on the periphery of inlet tube may suppress the boundary layer separation process in micro gas turbine. But practical application of this method can again be verified by using CFD studies in relation to number, size, shape and position of openings.

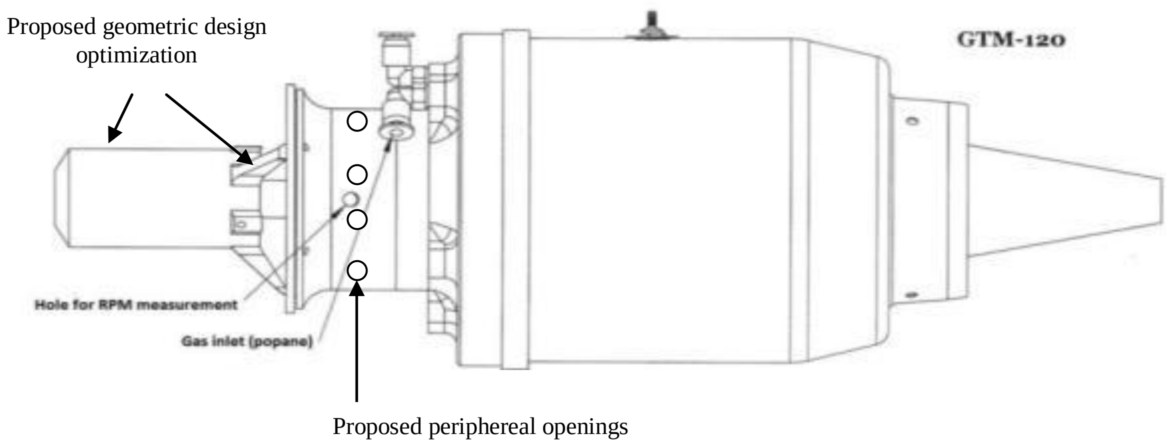


Fig10.1 Inlet geometric design propositions

10.2 Compressor

Due to its low cost, ease of manufacturing and design simplicity, centrifugal compressor has been the first choice of most commercial and amateur micro gas turbine manufacturers. Compression ratio of a typical micro class gas turbine ranges from 2-4, which has been a limiting factor in increased overall performance of system. Axial micro compressors on other hand suffer from decreased compression efficiency with decreasing dimensions with included additional weight, cost and complexity. Thus it can be concluded that within current technological scenario, centrifugal compressors are best suited for micro gas turbines.

10.3 Combustion chamber

In author's assessment of presented work, it will be right to conclude that combustion chamber is the area which offers tremendous scope of research on enhanced performance of engines at micro scale. Complexity of the combustion process which is further intensified by changing combustion dynamics at micro dimensions poses a greater challenge so as the vast research opportunities it offers.

It will not be an exaggeration to state that enhancing the efficiency of combustion process is the only method to gain practical and meaningful performance improvements in micro gas turbine. Low combustion efficiency and relatively high emissions from micro gas turbines can be attributed mainly to three factors:

- high heat loss through combustion chamber walls
- very less residence time of reactant species for combustion reaction to complete
- improper mixing of air and fuel due to smaller length of primary zone

10.3.1 Propositions

Due to high surface area to volume ratio of combustion chamber, micro gas turbine inherently suffers from high heat loss which highly influences combustion efficiency. Thermal Barrier Coatings, which are extensively used in standard gas turbine engines, are proposed here to limit the heat loss from micro combustor. Through modern day advanced coating techniques can provide a good quality of thermal barrier material coating which in author's opinion may lead to significant improvement in combustion efficiency due to reduced heat loss.

Trapped Vortex Combustion concept, which is studied in preliminary details in previous chapters, can be an effective solution method to deal with remaining two factors described above i.e. less residence time and improper mixing of reactant species. By injecting and burning the fuel in low velocity trapped vortex zone can potentially provide better mixing due to vortices and sufficient reaction time due to low flow velocity.

Combustion process initiates in fuel rich pilot zone and emanating hot gases sustain combustion of fuel injected in lean main zone with increasing power above idling. Relative amount of fuel fed in two zones must be such that no thrust lag is introduced. Together pilot and main zone have potential to maintain an optimal equivalence ratio needed to ensure lower NOx emissions at higher power output.

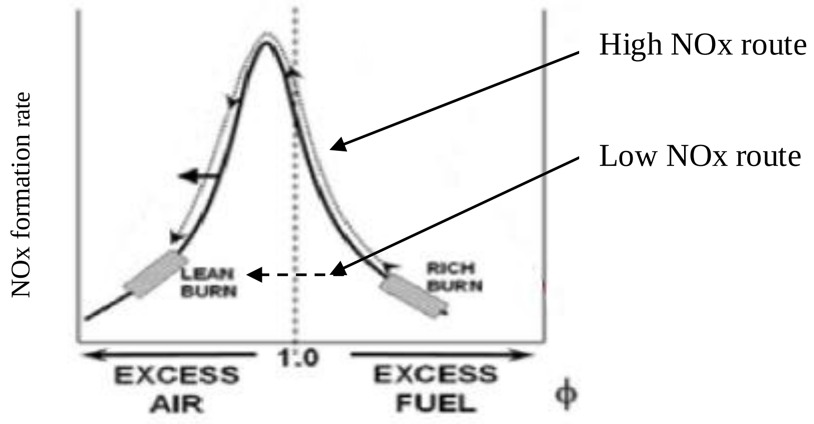


Fig 10.2 Graph for NOx formation rate dependence on equivalence ratio [2]

As shown in figure 10.2 above, using the proposed TVC combustion schematic, high NOx route can be avoided by combining pilot and main combustion zone as depicted in figure 10.3.

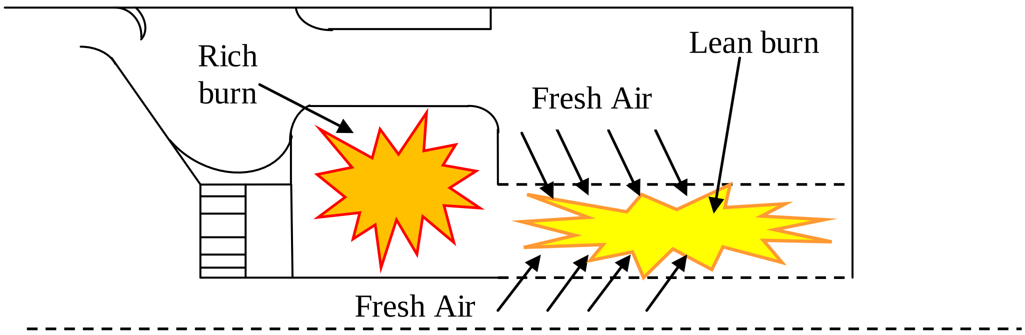


Fig 10.3 Proposed conception of combustion process organization

In proposed Trapped Vortex Combustion configuration for micro gas turbine, compressed air must be redirected using special casing for centrifugal compressor. This additional component may tend to increase weight, front area and overall length of the system. Gain in combustion stability, lower emissions and higher efficiency relative to increment in weight, cost and dimensions may act as a direction of future research on these lines.

Also, another dimension into the future research based on this study can be added by studying the influence on flow field by using a swirler into the inlet channel. Conventional swirlers such as vane swirlers are complex in design which may further increase overall length of engine and may not generate a stabilized swirl at micro scale. Author finds plate orifice swirler can be more suitable proposition for micro scale gas turbines.

10.4 Power Turbine and Nozzle

Single stage power turbine and fixed geometry nozzle has been sufficiently used for meeting current demands from micro gas turbine. Low cost, compactness and design simplicity are the major factors which make single stage turbine and fixed shape nozzle most suitable for current technological applications of micro gas turbines.

10.5 Final conclusions

- A step by step examination and analyses conducted in this study provides better understanding on influence of structural geometry of micro gas turbine engine parts on its overall performance.
- Based on this examination and analysis, it can be strongly suggested that significant gain in overall performance of a micro gas turbine engine is possible to attain by introducing optimal changes in its structural geometry, especially in that of combustion chamber.
- Key advanced combustion chamber geometries are examined and their feasibility at micro scale is studied in detail. Trapped Vortex Combustion (TVC) concept is found to be most feasible solution with high potential to enhance performance of a micro gas turbine engine.
- Based on CFD studies of simplified TVC geometry, a step ratio of 1.5 with heat source distributed uniformly on location T1 and T2 is found to generate most appropriate conditions for stable combustion process to occur. Step geometry must be designed to ensure generation of a stable vortex over full operating range of micro gas turbine engine.
- A detailed schematic is presented for a micro gas turbine engine with Trapped Vortex Combustion (TVC) chamber design.

11. References

[1] Mini- and micro-gas turbines for combined heat and power -P.A. Pilavachi, 2002

[2] Micro Gas Turbine Engine: A Review, "Progress in Gas Turbine Performance", book edited by Ernesto Benini, ISBN 978-953-51-1166-5, 2013, InTech Open

[3] Intermediate Master Project: “A comparison study of calculated gas dynamic parameters with that obtained from laboratory tests for micro turbo jet engine-GTM 120” Nikhi Madan, 2014, ITC-PW

[4] Federal Register- United States Government (https://www.federalregister.gov/articles/2013/10/23/2013-24712/exhaust-emissions-standards-for-new-aircraft-gas-turbine-engines-and-identification-plate-for#h-17)

[5] Beginner’s Guide to Propulsion, NASA Glenn Research Center (http://www.grc.nasa.gov/WWW/k-12/airplane/bgp.html)

[6] JETPOL - Polskie Silniki Serii GTM, www.jetpol.home.pl

[7] Gas Turbine Combustion-Alternative Fuels and Emissions by Arthur H. Lefebvre and Dilip R. Ballal, 3rd edition, CRC Press

[8] www.wikipedia.org

[9] Combustion Technology: Present and Future, TLC-Towards Lean Combustion EU’s priority 4 (Aeronautics and Space) program under 6th framework. (http://gfn.unizar.es/tlc/?q=background)

[10] “CFD Python: 12 Steps to Navier-Stokes” (http://lorenabarba.com/blog/cfd-python-12-steps-to-navier-stokes/)

[11] Computational Fluid Dynamics: An introduction, Wendt J.F. (ed.), Springer

[12] NPTEL- www.nptel.ac.in

[13] http://www.cfd-online.com/Wiki/Lid-driven_cavity_problem

[14] http://www.openfoam.org/archive/1.7.1/docs/user/cavity.php

[15] http://www.glassonweb.com/articles/article/854/ Flameless Oxidation as a means to reduce NOx emissions in glass furnace

[16] http://advtechconsultants.com/GasTurbineCombustion.htm

[17] www.grabcad.com

[18] plus.google.com

[19] http://www.dapi-tech.de/EN/turbine_cad_en.html

[20] fem.unicamp.br, pilotfriend.com

[21] http://direns.mines-paristech.fr/Sites/Thopt/fr/co/chambres-combustion.html

[22] www.rcuniverse.com

[23] globalspec.com