Analysis of Can Type Combustion Chamber – A Review

Naitik H. Gor[#1], Milan J. Pandya[*]

[#]PG Student (Thermal Engineering), Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.

[*]Assistant Professor, Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.

***Abstract*— The following review paper shows the different major analysis done by several researchers in the topic of Can Type Combustion Chamber. According to our review we find that majority of the researches used Computational fluid dynamics (CFD) modelling as combustion optimization tool. The steady increase in computer power over recent years has enabled combustion engineers to model reacting multi-phase flows in a realistic geometry with good mesh resolution. As a result, the number of applications of CFD to industries and power generation are also growing rapidly and increasing in sophistication. This paper reviews some of the recent applications of the CFD in gas turbine combustor used in the power generation, aero-engines and combustion industries. The aim is to illustrate what can be done and also to identify trends and those areas where further work is needed.**

Keywords*— Computational Fluid Dynamics, smoke rings, emission characteristics, swirl angle.

I. Introduction

The last 20 years has seen a large growth in Gas Turbine Technology. The growth is spearheaded by the growth of materials technology, new coatings and new cooling schemes. This, with the conjunction of increase in compressor pressure ratio, has increased the gas turbine thermal efficiency from about 15% to over 45%. It has been observed that the increase in the exhaust temperature of the combustor increases the gas turbine efficiency.

A. Basics of Can Type Combustion Chamber

Introduction of Combustion chamber

The combustion chamber (Fig.1) has the difficult task of burning large quantities of fuel, supplied through the fuel spray nozzles, with extensive volumes of air, supplied by the compressor and releasing the heating such a manner that the air is expanded and accelerated to give a smooth stream of uniformly heated gas at all conditions required by the turbine. This task must be accomplished with the minimum loss in pressure and with the maximum heat release for the limited space available. The amount of fuel added to the air will depend upon the temperature rise required. However, the maximum temperature is limited to within the range of 850 to 1700 deg C. by the materials from which the turbine blades and nozzles are made. The air has already been heated to between 200 and 550 deg. C. by the work done during compression, giving a temperature rise requirement of 650 to 1150 deg. C from the combustion process. Since the gas temperature required at the turbine varies with engine thrust, and in the case of the turbo-propeller engine upon the power required, the combustion chamber must also be capable of maintaining stable and efficient combustion over a wide range of engine operating conditions. Efficient combustion has become increasingly important because of the rapid rise in commercial aircraft traffic and the consequent increase in atmospheric pollution, which is seen by the general public as exhaust smoke.

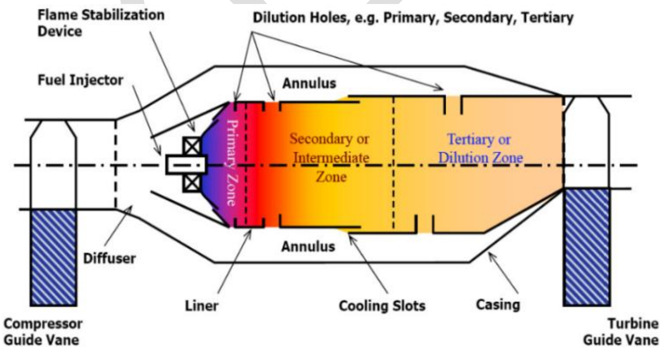

Fig. 1 Basic Construction of a Combustion Chamber.

Combustion Process in the Combustion chamber

Air from the engine compressor enters the combustion chamber at a velocity up to 500 feet per second, but because at this velocity the air speed is far too high for combustion, the first thing that the chamber must do is to diffuse it, i.e. decelerate it and raise its static pressure. Since the speed of burning kerosene at normal mixture ratios is only a few feet per second, any fuel lit even in the diffused air stream, which now has a velocity of about 80 feet per second, would be blown away. A region of low axial velocity has therefore to be created in the chamber, so that the flame will remain alight throughout the of a combustion chamber can vary between 45:1 and 130:1, However, kerosene will only burn efficiently at, or close to, a ratio of 15:1, so the fuel must be burned with only part of the air entering the chamber, in what is called a primary combustion zone. This is achieved by means of a flame tube (combustion liner) that has various devices for metering the air flow distribution along the chamber. Approximately 20 per cent of the air mass flow is taken in by the snout or entry section (Fig.2).Immediately downstream of the snout are swirl vanes and a perforated flare, through which air passes into the primary combustion zone. The swirling air induces a flow upstream of the centre of the flame tube and promotes the desired recirculation. The air not picked up by the snout flows into the annular Space between the flame tube and the air casing.

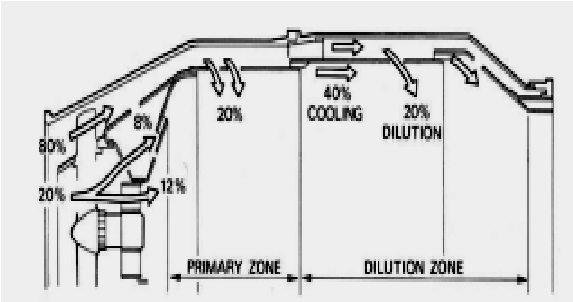

Fig. 2 Air Flow Distribution along the Combustion Chamber

Through the wall of the flame tube body, adjacent to the combustion zone, are a selected number of secondary holes through which a further 20 per cent of the main flow of air passes into the primary zone. The air from the swirl vanes and that from the secondary air holes interacts and creates a region of low velocity recirculation. This takes the form of a toroidal vortex, similar to a smoke ring, which has the effect of stabilizing and anchoring the flame (Fig.3).The re-circulating gases hasten the burning of freshly injected fuel droplets by rapidly bringing them to ignition temperature.

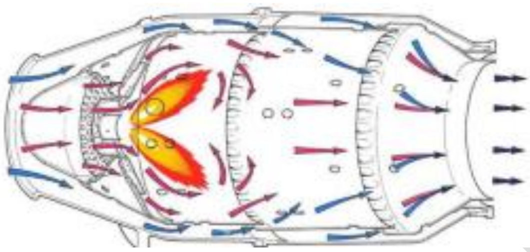

Fig. 3 Smoke Rings inside the Combustion Chamber

It is arranged such that the conical fuel spray from the nozzle intersects the recirculation vortex at its centre. This action, together with the general turbulence in the primary zone, greatly assists in breaking up the fuel and mixing it with the incoming air. The temperature of the gases released by combustion is about 1,800 to 2,000 deg. C, which is far too hot for entry to the nozzle guide vanes of the turbine. A recent development allows cooling air to enter a network of passages within the flame tube wall before exiting to insulating film of air. This can reduce the required wall cooling airflow by up to 50 per cent. Combustion should be completed before the dilution air enters the flame tube, otherwise the incoming air will cool the flame and incomplete combustion will result.

II. LITERATURE REVIEW ON CAN TYPE COMBUSTION CHAMBER

H. A. Bhimgade, S. K. Bhele, "A Review on use of Computational Fluid Dynamics in Gas Turbine Combustor Analysis and its Scope". [1]

In this paper the CFD application and its scope, is mainly focused on gas turbine combustor (generally can or tubular, annular and tub annular type of combustor used in gas turbines for higher efficiency). In many practical combustion applications like gas turbine and diesel engine, the combustion takes place in turbulent flow field. Therefore it is important to model the effects of turbulence and mixing interactions including all related processes either physical or chemical. In the present the emphasis is on how the turbulence leads to increased mixing in order to be used to compensate for the inaccurate prediction for the chemical reaction rate. However this has to be treated numerically and physically. Both ways are referring to the incomplete mixing process that may lead to ignite the fuel vapour before the auto-ignition delay time or out of the main reaction zone. Physically, the mixing process tends to speed up the overall reaction rate by stretching and wrinkling of the preheating zone. In addition the simulation of turbulent spray combustion remains quite a hard task because many problems may occur due to strong coupling that exists between predicted vapour mass fraction and the chemical reaction.. The experimental results and the semi empirical correlations for calculating CO, UHC, NOx, exhaust gases temperature and inner liner wall temperature as a function of different operating parameters are useful for design and further development of design, of gas turbine combustor is possible. Even with existing physical models, CFD can offer cost-effective solutions for many complex systems of interest to the power generation, aero-engines and process industries.

Sachin Bhalerao., Dr. A.N.Pawar. "Thermal mapping of a can type gas turbine Combustion chamber using CFD". [2]

This paper reveals that the increase in the exhaust temperature of the combustor increases the gas turbine efficiency. In this paper, simulation of the thermal flow behavior of the combustion chamber using CFD and their results are discussed. The combustor understudy is a typical can type combustion chamber. The overall length of combustor is 567mm including the diffuser section at outlet. A mesh of 557K nodes is designed after a set of trials which showed that further refinement in either direction does not change the velocity and scalar variables at any point in the combustor considerably. The domain is discretized with 3 million full tetrahedral volume cells. The grids are generated by a mesh generator. 2-D quadrilateral grids are chosen to approximate the domain. Fig. 4(a), 4(b) and 4(c) shows the different views of the meshed geometry. Fig. 5 shows the temperature distribution along the length of the combustion chamber.

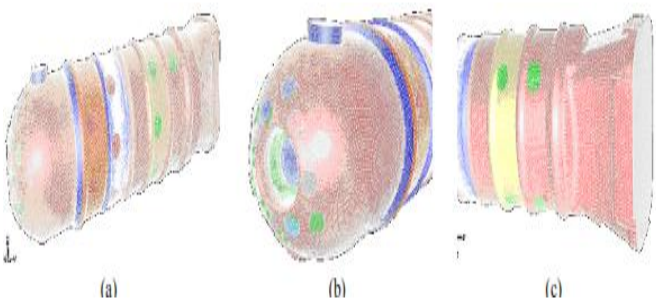

Fig. 4 Different Views of meshed geometry for a Combustion Chamber

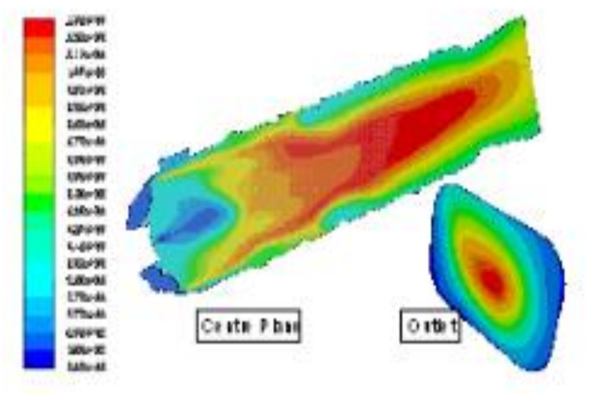

Fig. 5 Temperature Distribution along the length of Combustion Chamber

Selvakuma Kumaresh, Man Young Kim, “Combustion and Emission Characteristics in a Can-type Combustion Chamber”. [3]

In this study, different flow configurations of various swirl angles are compared and motility of holes in the secondary chamber are altered to examine about the emission of unburned gases and to obtain effective combustor with less NOx emission. Numerical investigation on Can-type combustion chamber shows that 60° swirl geometry is giving less NO emission as the temperature at the exit of combustion chamber is less as compared to 30° and 45° swirl angle geometry.

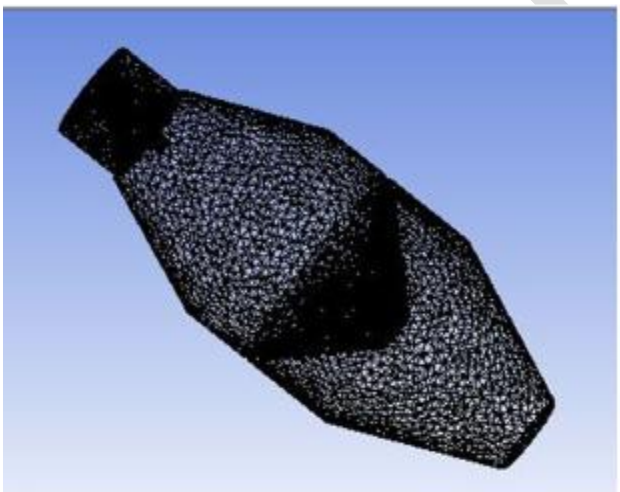

Fig. 6 Mesh Domain for a Can Combustor

Fig. 6 shows the mesh systems of can combustor adopted in this work. An algebraic grid-generation technique is employed to discretize the geometry inside the computational domain. NOx emission. Fig. 7 demonstrates the NOx mass fraction for the can-combustor with different swirl angles and axial distance. It is evident from the figure that after the location of primary chamber at Xs=0.7, the emission of NOx diminishes because of the cooling effect of air. Hence, it can be stated that the 60 degree swirl angle attempts less NOx emission due to low exit temperature.

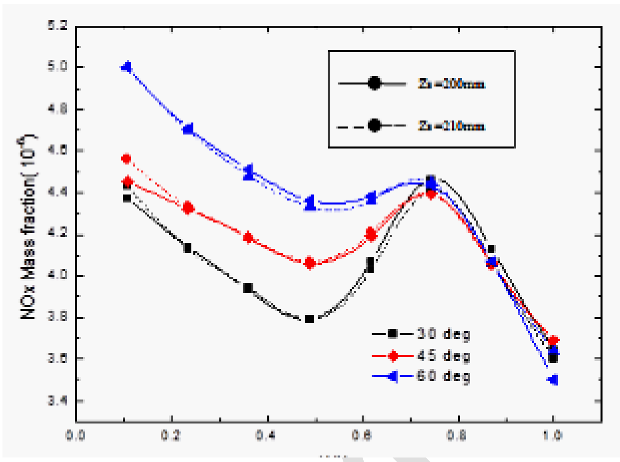

Fig. 7 Variations of NOx mass fraction for the can-combustor with different swirl angles and axial distance.

The results from the parametric studies indicate that the calculation of NOx emission serves to develop low emission combustor. Thus, it can be noted that the use of 60 degree axial swirl angle at the secondary inlet location of 200mm achieve slow emission and to prevail better performance in aerospace applications.

P. Sravan Kumar, P. Punna Rao, “Design and Analysis of Gas Turbine Combustion Chamber”. [4]

This paper presents the design of combustion chamber followed by three dimensional simulations to investigate the velocity profiles, species concentration and temperature distribution within the chamber and the fuel considered as Methane (CH4). According to their analysis, the following conclusions are derived.

The static temperature is very high in the regions where combustion takes place and goes on decreasing towards the outlet. The maximum temperature reached is 2500 K which indicates that there is efficient combustion process. The turbulent intensity is high in the immediate vicinity of the ramp injector indicating a superior air-fuel mixing. A very high turbulent intensity indicates a superior air-fuel mixing. The sudden rise in temperature observed near the tip of the injector indicates the generation of shocks which help in superior air-fuel mixing. Superior air-fuel mixing resulting in better quality of combustion and thus better performance.

P.S.Jeyalaxmi, Dr.G.Kalivarathan, “CFD analysis of flow characteristics in a gas turbine- A viable approach to predict the turbulence”. [5]

In this investigation it is observed that the RNG k-ε turbulence model used in the cases is viable and it is seen that the RSM model remarkably exhibits the turbulence level and the dilution jet mixing at the combustor outlet. But still, It is easier to predict the variations through CFD in the mean flow field results. Because of these reasons more efforts has to be made towards testing additional turbulence models to access one more suitable means for predicting this type of swirling, highly turbulent and flow situation in a generic manner through viable CFD tool.

Ramazan, Jawaz Pasha. , Abdul Mujeeb M S, “CFD Simulation of Swirling Effect in S-Shaped Diffusing Duct by Swirl Angle of 10”. [6]

This paper involves the CFD analysis for the prediction of swirl effect on the characteristics of a steady, incompressible flow through an S-shaped diffusing duct by keeping swirl angle of 10 degree. The curved diffuser considered in the present case has S-shaped diffusing duct having an area ratio of 1.9, length of 300 mm and turning angle of 22.5 deg.

On the basis of analysis and results presented, the following conclusion can be drawn. The overall static pressure recovery increases for swirl flow at inlet irrespective of direction of swirl (clockwise or anticlockwise), the increase being around 40%. The flow distribution at the exit plane is more uniform for clockwise swirl in comparison to anti – clockwise swirl. The secondary flow is present throughout the diffuser for uniform flow. A pair of vortices is observed at all the sections. For swirl flow, only circular motion is present and no pair of vortices is formed. Turbulence intensity variation indicates the presence of high turbulence intensity at the planes just before and after the inflexion plane. Turbulence intensity variation at the exit for the clockwise swirl flow is much lower compared to the uniform flow at the inlet. Fig 8(a) shows turbulence intensity distribution with the clockwise swirl flow and Fig 8(b) shows turbulence intensity distribution with the anticlockwise swirl flow.

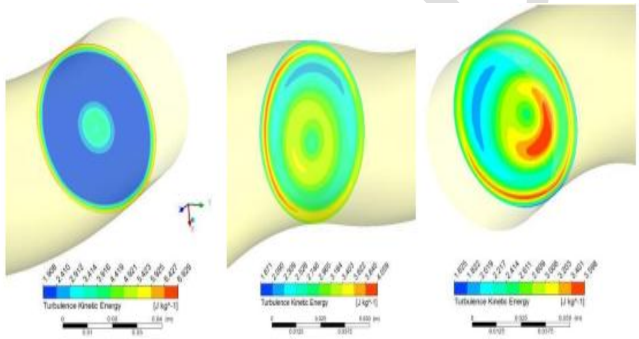

Fig. 8(a) Turbulence intensity distribution with the clockwise swirl flow

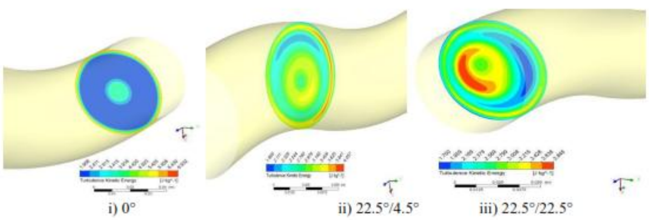

Fig. 8(b) Turbulence intensity distribution with the anticlockwise swirl flow

III. CONCLUSION

After analysing and understanding all the above literatures, we can conclude that by optimization we can increase the combustion efficiency for a can type combustion chamber. Using CFD simulation, the effective design for such type of combustion chamber can be developed. After making some of the design modifications, we can minimize the pressure loss and can maximize heat release for limited space available. Using some of the experimental results and the semi empirical correlations for calculating CO, UHC, NO, exhaust gas temperature, inner liner wall temperatures, etc. developments in the design for a can type combustion chamber is possible. Such design modifications can offer cost-effective solutions for many complex systems of interest to the power generation, aero-engines and process industries.

REFERENCES

[1] H. A. Bhimgade, S. K. Bhele, A Review on use of Computational Fluid Dynamics in Gas Turbine Combustor Analysis and its Scope, International Journal of Science and Research (IJSR), India Online ISSN: 2319-7064.

[2] Sachin Bhalerao., Dr. A.N.Pawar, Thermal mapping of a can type gas turbine combustion chamber using CFD, International Journal of Emerging trends in Engineering and Development. ISSN 2249-6149 Issue2, Vol.1.

[3] Selvakuma Kumaresh, Man Young Kim, Combustion and Emission Characteristics in a Can-type Combustion Chamber, International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering Vol:8 No:7/ 2014.

[4] P.Sravan Kumar, P.Punna Rao, Design and Analysis of Gas Turbine Combustion Chamber, International Journal Of Computational Engineering Research, Vol. 03 Issue. 12/2013.

[5] P.S.Jeyalaxmi, Dr.G.Kalivarathan, CFD analysis of flow characteristics in a gas turbine-A viable approach to predict the turbulence, International Journal of Mechanical Engineering and Technology (IJMET), ISSN 0976, Volume 4, Issue 2, March - April 2013.

[6] Ramazan , Jawaz Pasha. , Abdul Mujeeb M S, CFD Simulation of Swirling Effect in S-Shaped Diffusing Duct by Swirl Angle of 10, IOSR Journal of Mechanical and Civil Engineering, ISSN: 2320-334X, Volume 6, Issue 2, Mar. - Apr. 2013.

[7] R.K.RAJPUT, Internal combustion engines:(including air compressors and gas turbines and jet propulsion), Bangalore, India, Laxmi Publications, 2007.

[8] V. Ganesan, Gas Turbines 3E, Tata McGraw-Hill Education, 01 Apr 2010.

[9] John D. Anderson, Computational Fluid Dynamics Jr., McGraw – Hill Education (India) Private Limited.