

Combustion and Emission Characteristics in a Can-type Combustion Chamber

Selvakuma Kumaresh, Man Young Kim

Abstract—Combustion phenomenon will be accomplished effectively by the development of low emission combustor. One of the significant factors influencing the entire Combustion process is the mixing between a swirling angular jet (Primary Air) and the non-swirling inner jet (fuel). To study this fundamental flow, the chamber had to be designed in such a manner that the combustion process to sustain itself in a continuous manner and the temperature of the products is sufficiently below the maximum working temperature in the turbine. This study is used to develop the effective combustion with low unburned combustion products by adopting the concept of high swirl flow and motility of holes in the secondary chamber. The proper selection of a swirler is needed to reduce emission which can be concluded from the emission of Nox and CO₂. The capture of CO₂ is necessary to mitigate CO₂ emissions from natural gas. Thus the suppression of unburned gases is a meaningful objective for the development of high performance combustor without affecting turbine blade temperature.

Keywords—Combustion, Emission, Can-type Combustion Chamber, CFD, Motility of Holes, Swirl Flow.

I. INTRODUCTION

THE combustor in a gas turbine is to add energy to the system to power the turbines, and produce high velocity gas to exhaust through the nozzle in aircraft applications. Combustion chambers must be designed to ensure stable combustion of the fuel injected and optimum fuel utilization within the limited space available and over a large range of air/fuel ratios. In a gas turbine engine, the combustor is fed by high pressure air by the compression system. A combustor must contain and maintain stable combustion despite very high air flow rates. To do so combustors are carefully designed to first mix and ignite the air and fuel, and then mix in more air to complete the combustion process.

To be more competent, the combustor model is chosen from the literature work of Reddy and Kumar [1]. This paper presents the experimental and numerical results for a two stage combustor capable of achieving flameless combustion. The concept of high swirl flows has been adopted to achieve high internal recirculation rates in flameless combustion mode. Computational analysis of the flow features shows that decrease in the exit port diameter of the primary chamber increases the recirculation rate of combustion products and

helps in achieving the flameless combustion mode. Detailed experimental investigations show that flameless combustion mode was achieved with evenly distributed combustion reaction zone and uniform temperature distribution in the combustor. The preference of natural gas is chosen from this investigation. Industrial gas turbines have a wider scope of fuel. Ghenai [2] has done numerical investigation of the combustion of syngas fuel mixture in gas turbine can combustor to understand the impact of the variability in the alternative fuel composition and heating value on combustion performance and emissions. The composition of the fuel burned in can combustor was changed from natural gas (methane) to syngas fuel with hydrogen to carbon monoxide (H₂/CO) volume ratio ranging from 0.63 to 2.36. Results show the changes in gas turbine can combustor performance with the same power generation when natural gas or methane fuel is replace by syngas fuels. The gas temperature for the all five syngas shows a lower gas temperature compared to the temperature of methane. The gas temperature reduction depends on lower heating value and the combustible and non-combustible constituents in the syngas fuel which results in less emission.

As a result some knowledge about utilization of Methane is carried out. Kumar and Rao [3] analyzed the gas turbine combustion chamber based on combined theoretical and empirical approach. The investigation of velocity profiles, species concentration and temperature distribution within the chamber and the fuel considered as Methane (CH₄) are carried out. The computational approach attempts to strike a reasonable balance to handle the competing aspects of complicated physical and chemical interactions of the flow. The modeling employs non-orthogonal curvilinear coordinates, second order accurate discretization, tetra grid iterative solution procedure and SST turbulence model. Accordingly, in present study an attempt has been made through CFD approach to analyze the flow pattern with in combustion and through air admission holes and from these the temperature distribution in the chamber walls as well as the temperature quality at the exit of combustion chamber is obtained.

The combustion of methane-air mixture in gas turbine can-type combustion chamber was experimented numerically by Pathan et al. [4]. The objective is to understand the combustion phenomena and resulted emissions. With the high cycle temperature of modern gas turbine, mechanical design remains difficult and a mechanical development program is inevitable. The rapidly increasing use of computational fluid dynamics (CFD) in recent years has had a major impact on the design process, greatly increasing the understanding of the complex flow and so reducing the amount of trial and error

Selvakumar Kumaresh, Graduate Student, is with the Department of Aerospace Engineering, Chonbuk National University, Jeonju, Chonbuk 561-756, Republic of Korea

Man Young Kim, Professor, is with the Department of Aerospace Engineering, Chonbuk National University, Jeonju, Chonbuk 561-756, Republic of Korea (Corresponding Author, e-mail: manykim@jbnu.ac.kr).

required. The gas turbine can-combustor is designed to burn the fuel efficiently, reduce the emissions, and lower the wall temperature. In this study various parameters like air-fuel ratio, swirler angle of primary air inlet, axial position of dilution holes are changed to investigate the effect of these parameters on combustion chamber performance and emissions. In this study the mathematical models used for combustion consist of the PDF flamelet model and eddy dissipation combustion model for non-premixed gas combustion. The outcome of the work will help in finding out the geometry of the combustion chamber which will lead to less emission.

The flow visualization technique was demonstrated by Koutmos and McGuirk [5] in can-combustor model. They experimented a benchmark study on the investigation of the three-dimensional flow field water model. Flow visualization demonstrated that internal flow patterns simulated closely those expected in real combustors. The combustor comprised a swirl driven primary zone, annulus fed primary and dilution jets and an exit contraction nozzle. LDA measurements of the three mean velocity components and corresponding turbulence intensities were obtained to map out the flow development throughout the combustor. Besides providing information to aid understanding of the complex flow events inside combustors, the data are believed to be of sufficient quantity and quality to act as a benchmark test case for the assessment of the predictive accuracy of computational models for gas-turbine combustors. Aside from the flow visualization study, Eldrainy et al. [6] examined the flow inside the combustor. The flow field inside the combustor is controlled by the liner shape and size, wall side holes shape, size and arrangement (primary, secondary, and dilution holes), and primary air swirler configuration. Air swirler adds sufficient swirling to the inlet flow to generate central recirculation region which is necessary for flame stability and fuel air mixing enhancement. Therefore designing an appropriate air swirler is a challenge to produce stable, efficient and low emission combustion with low pressure losses. The flow behavior was investigated numerically using CFD solver. This study has provided physical insight into the flow pattern inside the combustion chamber.

The necessity of the problem is to evaluate the NO_x emission after the combustion process. Wunning and Wunning [7] carried out the experiments on flameless oxidation. In contrast to the combustion within stabilized flames, temperature peaks can be avoided at flameless oxidation. For that reason, the thermal NO-formation is largely suppressed even at very high air preheat temperatures. A brief summary of the present NO_x reducing techniques is given with the illustration of flameless oxidation. The results are encouraging that NO-emissions from a wide range of combustion sources could be largely eliminated in the future. Use of burners, operated in flameless oxidation mode in continuous industrial furnaces has proven to be reliable and well accepted for the very uniform product quality by furnace people. The temperature of the combustion process and peak values can be reduced by the provision of some excess air.

It is well known that the accurate evaluation of emission of unburned gases is inevitable for both efficient and clean

combustion. There are strict regulations on aircraft emissions of pollutants like carbon dioxide and nitrogen oxides, so combustors need to be designed to minimize those emissions. In this analytical work, an examination about the emission of unburned gases is predicted by motility the holes in the secondary chamber. The main objective of this study is to diminish the NO_x emission and to establish the effective swirler angle by altering the secondary holes for cooling the combustion products. Substantiality of flameless combustion by altering the location of secondary holes is observed for all three different swirler angles.

II. MATHEMATICAL MODELS

Fig. 1 depicts the typical two stage combustor configuration comprised of primary and secondary chambers. In this study, three different flow configurations of swirl angles of 30, 45 and 60 degrees are examined to study about the emission of unburned gases. Air is injected through centrally mounted pressure swirl injector containing 16 vanes, liquid fuel is injected through five normal injection ports and air from the secondary chamber are introduced by five normal injection ports for cooling the combustion products. The parameters represented in the Fig. 1 are length of whole combustion chamber of $X_s = 340$ mm, length of primary combustion chamber with $X_s = 220$ mm, $X/X_s=0$ (Inlet of primary combustor), $X/X_s=0.7$ (Outlet of primary combustor) and $X/X_s=1$ (Outlet of main combustor), while Z_s points the motility of holes in the secondary chamber which are analyzed for different positions at 200 and 210mm towards the downstream flow direction. Here, motility represents the change in position that doesn't entail a change of location.

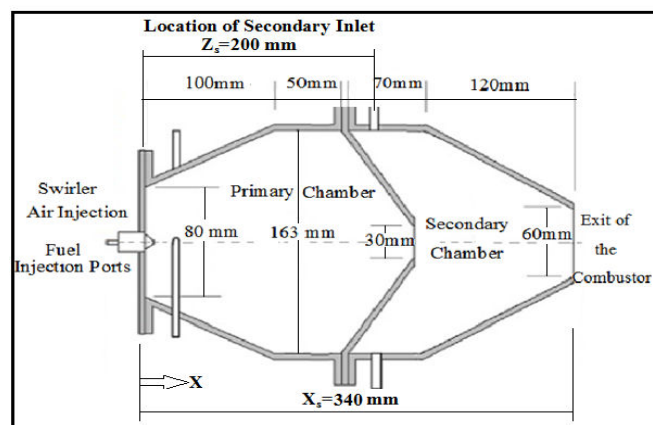


Fig. 1 Schematic diagram of the can-type combustor chamber model

In this numerical study, a fully implicit finite volume scheme of the Navier-Stokes equations is employed. Numerical simulations have been carried out using Fluent 14 under CFX analysis system. A three-dimensional k-epsilon turbulence and eddy dissipation combustion models are exercised. Also, methane-air mixture was chosen as a fuel-air mixture and P1 radiation model was attached to include the effect of thermal radiation on combusting flow field. We have chosen methane-air as a fuel air mixture and thermal radiation P1 model for

numerical studies. The investigation of velocity profiles, species concentration and temperature distribution are predicted within the chamber.

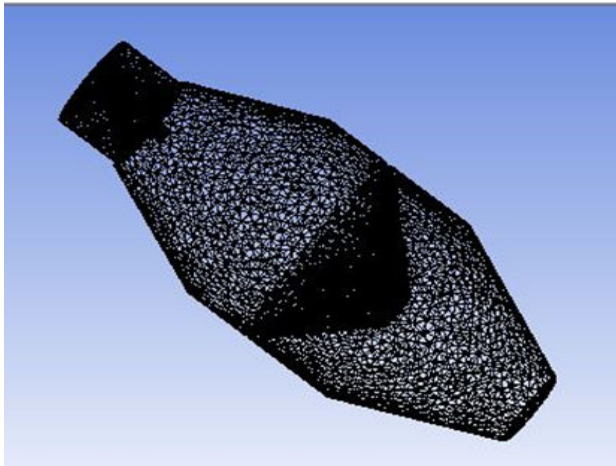


Fig. 2 Mesh domain of the can-combustor

The governing mass, momentum, energy, and species conservation equations are

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

$$\frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_{eff} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (2)$$

$$\frac{\partial}{\partial x_j}(\rho u_j h) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\frac{\mu_{eff}}{Pr_T} \frac{\partial h}{\partial x_j} \right] + H_v \dot{\omega}_f - \nabla \cdot q^R \quad (3)$$

$$\frac{\partial}{\partial x_j}(\rho u_j Y_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\frac{\mu_{eff}}{Sc_T} \frac{\partial Y_i}{\partial x_j} \right) + \dot{\omega}_i \quad (4)$$

where ρ , u , h , and Y are the density, velocity, enthalpy, and species mass fraction, respectively. Also, $H_v \dot{\omega}_f$ and $-\nabla \cdot q^R$ are the source terms due to combustion of fuel and thermal radiation, while, $\mu_{eff} = \mu + \mu_T$ and $\dot{\omega}_i$ are the viscosity and rate of production of species i , respectively. Here, turbulent viscosity is $\mu_T = C_\mu \rho k^2 / \varepsilon$. The turbulent kinetic energy (TKE) and its dissipation rate are computed from

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left(\frac{\mu_{eff}}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k - \rho \varepsilon \quad (5)$$

$$\frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left(\frac{\mu_{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_1 \frac{\varepsilon}{k} G_k - C_2 \rho \frac{\varepsilon^2}{k} \quad (6)$$

where G_k is the rate of the production of TKE, and related constants are obtained from Launder and Spalding [8] as $C_1 = 1.44$, $C_2 = 1.92$, $C_\mu = 0.09$, $C_k = 1.3$, and $C_\varepsilon = 1.0$. The detailed information for combustion and radiation are can be found in Kim [9].

Commercial S/W Fluent provides a turbulence-chemistry

interaction model called the eddy-dissipation model. It is usually used when turbulent mixing of the constituents has to be taken into consideration. Also, this model could well predict the uniform temperature variation at the exit. Therefore, it has been used for demonstrating the flow physics of combustor successfully during past few decades. The k-epsilon model is the transport model capturing the shear stress distribution and specific dissipation rate inside the combustor model. Thermal radiation P1 model is utilized solves the diffusion equation and radiation flux. In order to capture the flow physics, the transport equations of chemical compounds are also simulated.

III. RESULTS AND DISCUSSION

Fig. 2 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. The domain contains approximately 300,000 nodes and 1.72 million elements with the element size of 1.2mm. The requirement of excess air is necessary to assure the oxidation of hydrocarbon molecules. Using 10% of excess air, it is found that 18.838 kg of air is required to burn 1kg of fuel methane.

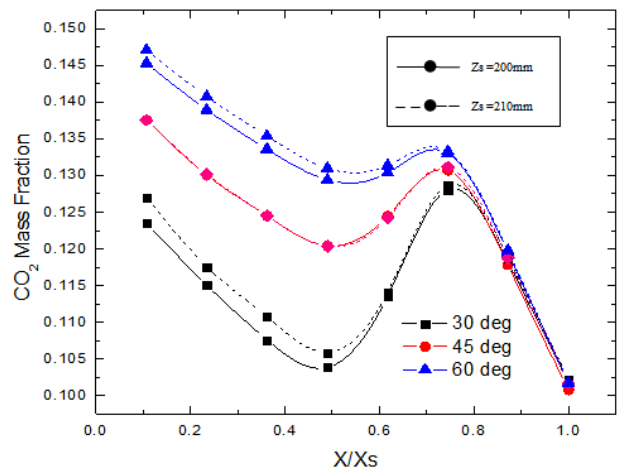


Fig. 3 CO₂ mass fraction for the can-combustor with different swirler angles and axial distance

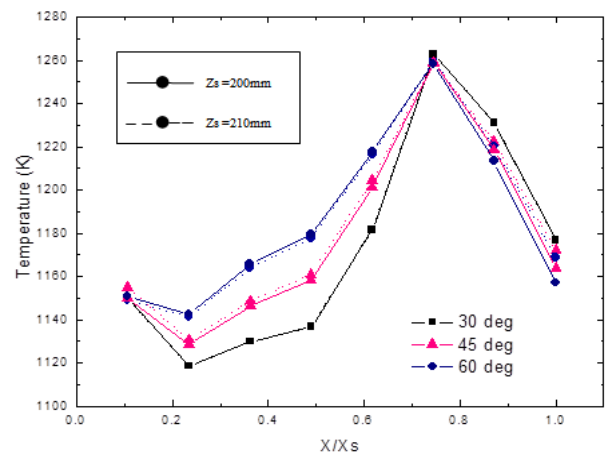


Fig. 4 Temperature for the can-combustor with different swirler angles and axial distance

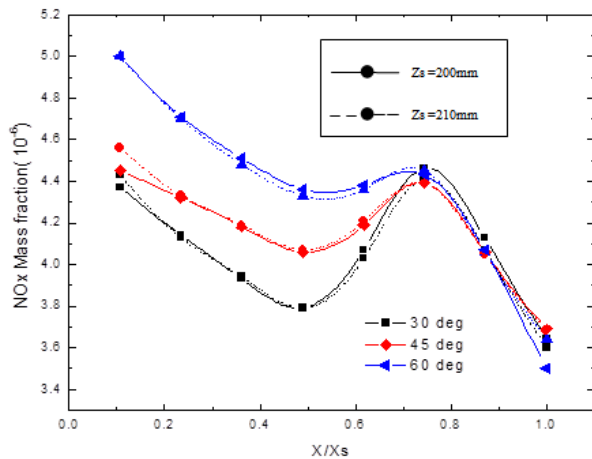


Fig. 5 NOx mass fraction for the can-combustor with different swirler angles and axial distance

The excess air injection is necessary to reduce the emission of NOx such that it shouldn't affect the efficiency of the combustion chamber and turbine inlet temperature. The proper selection of a swirler is required to reduce the emission which can be concluded from the emission of NOx and CO2. Fig. 3 demonstrates the CO2 mass fraction for the can-combustor with different swirler angles and axial distance. The air at the secondary inlet is introduced to reduce the NOx emission and also to mitigate CO2 emission from natural gas. In Fig. 3, it is clearly predicted that CO2 mitigated to the value of 10.5% due to the cooling effect of air from the secondary inlet by swirling velocity.

One of the thermodynamic parameter called temperature influences the flow inside the combustion chamber. To prognosticate this parameter, the selection of better swirler angle with respect to the location of secondary inlet is necessary. Fig. 4 demonstrates the temperature for the can-combustor with different swirler angles and axial distance. The temperature increases gradually due to the chemical reaction inside the main combustor. It is clear that after the location of primary chamber at $X_s=0.7$, the temperature diminishes due to the cooling effect of air from the secondary holes. Thus, it is distinct to show that the 60 degree swirler angle achieves better temperature in comparison with other swirler angles. Also, it can be found that at the location of $Z_s=200\text{mm}$, the temperature is diminished finer.

Depending upon the temperature the NOx emission is determined by the species equation. The main intention for introducing more air in the secondary chamber is to reduce the NOx emission. Fig. 5 demonstrates the NOx mass fraction for the can-combustor with different swirler angles and axial distance. It is evident from the figure that after the location of primary chamber at $X_s=0.7$, the emission of NOx diminishes because of the cooling effect of air. Hence, it can be stated that the 60 degree swirler angle attempts less NOx emission due to low exit temperature. Detailed experimental investigations show that flameless combustion mode can be achieved with evenly distributed combustion reaction zone and uniform temperature distribution in the combustor. At the location

$Z_s=200\text{mm}$, however, in secondary chamber serves better performance in compare with other swirler angles.

IV. CONCLUSION

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° swirler geometry is giving less NO emission as the temperature at the exit of combustion chamber is less as compared to 30° and 45° swirler angle geometry. So that for further numerical analysis 60° geometry is used. Temperature profiles shows increment at reaction zone due to burning of air-methane mixture and decrement in temperature at the downstream of secondary inlet holes due to supply of more to dilute the combustion mixture.

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 swirler angle at the secondary inlet location of 200mm achieves low emission and to prevail better performance in aerospace applications.

ACKNOWLEDGMENT

This work was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (grant number 2011-0022179). Also, this research was partially supported by Leading Foreign Research Institute Recruitment Program through the National Research Foundation of Korea funded by the Ministry of Education, Science and Technology (2011-0030065).

REFERENCES

- [1] V. M. Reddy and S. Kumar, "Development of high intensity low emission combustor for achieving flameless combustion," *Propulsion and Power Research*, Vol. 2, 2013, pp. 139-147.
- [2] C. Ghenai, "Combustion of syngas fuel in gas turbine can combustor," *Advances in Mechanical Engineering*, Vol. 1, 2010, pp. 1-13.
- [3] P. S. Kumar and P. P. Rao, "Design and analysis of gas turbine combustion chamber," *International Journal of Computational Engineering Research*, Vol. 3, 2012, pp. 1-6.
- [4] H. Pathan, K. Partel, and V. Tadvi, "Numerical investigation of the combustion of methane air mixture in gas turbine can-type combustion chamber," *International Journal of Scientific & Engineering Research*, Vol. 3, No. 10, 2012, pp. 1-7.
- [5] P. Koutmos and J. J. McGuirk, "Isothermal flow in a gas turbine combustor—a benchmark experimental study," *Experiments in Fluids*, Vol. 7, 1989, pp. 344-354.
- [6] Y. A. Eldrainy, J. Jeffrie, and M. Jaafar, "Prediction of the flow inside a Micro Gas Turbine Combustor," *Journal of Mechanical*, vol. 25, 2008, pp. 50-63.
- [7] J. A. Wunning, and J. G. Wunning, "Flameless oxidation to reduce thermal NO-formation," *Progress in Energy and Combustion Science*, Vol. 23, No. 1, 1997, pp. 81-94.
- [8] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, 1974, pp. 269-289.
- [9] M. Y. Kim, "Effect of swirl on gas-fired combustion behavior in a 3-D rectangular combustion chamber," *World Academy of Science, Engineering and Technology*, Vol. 64, 2012, pp. 939-944.