

Numerical Investigation of the Combustion of Methane Air Mixture in Gas Turbine Can-Type Combustion Chamber

Firoj H Pathan, Nikul K Patel, Mihir V Tadvi

Abstract— three dimensional numerical investigation of the combustion of methane air mixture in gas turbine Can- combustor by using CFD with CFX solver is presented in this study. 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 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 Flame let 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.

Index Terms— Combustion, CFX, Emission, Eddy Dissipation Combustion Model, Gas Turbine , Methane Air mixture combustion, PDF Flame let Model.

1 INTRODUCTION

THE combustion chamber has the difficult task of burning large quantities of fuel, supplied through the fuel burners, with extensive volume of air, supplied by the compressor, and releasing the heat in 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. Generally the air-fuel ratio in open gas turbines varies from 50:1 to 200:1 to get efficient combustion and to keep the turbine inlet temperature down to permissible limits. The design of a gas turbine combustion system in a complex process involving fluid dynamics, combustion and mechanical design is complex. For many years the combustion system was much less amenable to theoretical treatment than other components of the gas turbine, and any development program required a considerable amount of trial and error. With the high cycle temperature of modern gas turbine, mechanical design remains difficult and a mechanical development program is inevitable. Atmospheric condition changes quite rapidly during climb and descent and combustor chamber has to meet all operational requirements. Industrial gas turbines have a wider scope of fuel. Natural gas is most preferred.

Chaouki Ghenai [1] 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 (H_2/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 to less emission. McGuirk and Palma [2] carried out experiments for flow inside the water model of a gas turbine combustor, with the two main objectives of increasing the understanding the type of flow and providing experimental data to assist the development of mathematical models. The main features of the geometry are the interaction between two rows of radially opposed jets penetrating a cross-flowing axial stream with and without swirl, providing a set of data of relevance to all flows containing these features. This helps in understanding the flow pattern during combustion phenomena. Jaafar et. al. [3] investigated the flow pattern in a gas turbine combustion chamber by simulation and experimental approaches and found highest swirl number of 2.29 for flat vane and 1.57 for curve vane. They are capable of creating a clear reversal mass flow rate zone and higher swirl strength reduces the corner recirculation zone size and hence reduces the negative impact on the combustion process and the homogeneity of the wall temperature as well. From the parametric study carried out with varying angle of vanes, it was found that 50° swirler is the best for producing appropriate recirculation zone with reasonable pressure drop. Koutmos and McGuirk [4] conducted experiments

- Firoj H Pathan is currently pursuing masters degree program in mechanical engineering in The M S University of Baroda, India, PH-0919913557457. E-mail: firojpathan_07@yahoo.com
- Nikul K Patel is currently working as Assistant Professor in mechanical engineering in The M S University of Baroda, India, PH-0919429926405. E-mail: nikulatmsu@gmail.com
- Mihir V Tadvi is currently working as Assistant Professor in mechanical engineering in the M S University of Baroda, India, PH-0918264353836, E-mail: mihir_tadvi2002@yahoo.com

for investigation of swirler/dilution jet flow split on primary zone flow patterns in a water model can-type combustor. Flow visualization studies revealed that significant changes occur in this region due to the interaction between the swirling flow and the radially directed primary jets. A large toroidal recirculation was formed and high levels of turbulence energy were generated in the core of the combustor at low levels of swirler flow rate. As the swirl level increases, the strength of this recirculation was observed to weaken. Beyond a critical level, the primary recirculation was pushed off centre and the undesirable feature of a forward velocity on the combustor axis in the primary zone was observed. Sokolov et. al. [5] studied a mathematical model for the description of ax symmetric swirled flow with diffusion combustion is based on numerical solution of the Reynolds equation with a $k-\omega$ model of turbulence. The results of numerical and experimental investigations of local and general characteristics of flow, heat and mass transfer, combustion, and NOx formation in an annular combustor with opposite swirled air jets are presented. Satisfactory agreement between numerical calculations and experiments is obtained which shows that the dependencies of combustor characteristics versus geometry and operational parameters are generalized. Kulshreshtha, et. al. [6] carried out numerical simulation of the designed can-annular type combustion chamber for small gas turbine engine using commercial CFD code CFX. The exit temperature quality is not uniform, this may be due to very low air flow rates and secondly the intermediate zone is neglected. To overcome this lacuna, a mixing length concept was introduced, which resulted in giving near uniform temperatures at the exit. Tomczak et. al. [7] had carried out the investigation of a gas turbine combustion system fired with mixtures of natural gas and hydrogen. The aim of this work was to evaluate with numerical and experimental methods, the performances on a gas turbine combustor fed with hydrogen rich mixtures without any changes of the general combustion system configuration. Increasing the share of hydrogen, a considerable changing of the flame shape could be noticed. The results were showing that the hydrogen flame was more compact, shorter and the flame temperatures were higher. While the largest hottest zone of burning natural gas was right in front of the dilution zone, the region of highest temperatures of burning hydrogen was moving upstream to the front part of the combustor (direction of gas inlet). The values of the highest temperature while burning pure hydrogen were about 2330 K and while using natural gas about 2290 K. Furu-hata et. al. [8] carried out experiment on low NOx combustor for kerosene-fueled micro gas turbine based on a new concept was proposed and the combustion characteristics of the prototype combustor were investigated. The new concept combustor consisted of primary and secondary combustion zones, and they were connected by a throat. A swirler was set between the primary and secondary combustion zones. In order to enhance the recirculation of burned gas in the primary combustion zone, the combustion air was introduced through the swirler and forced to flow upward to the combustor bottom, from where fuel spray was supplied through a nozzle. An optimum configuration of the primary combustion zone such as length of primary zone, swirler vane angle, diameter of throat, etc. were investigated to achieve high combustion stability and low emis-

sion in wide ranges of fuel flow rate and excess air ratio. Benini et. al. [9] carried out Numerical and experimental investigations to assess the benefits and drawbacks of both water (mist) and steam direct injection within the combustion chamber of a 200 N static thrust turbojet. The aim of the investigations is to evaluate the impact of increasing water and steam flows (ranging from 0% to 200% of the fuel mass flow) onto the emissions levels (NO and CO) of the engine. Experimental and numerical analyses show that in this particular turbojet, steam and water injections permit reduction in NO emission and confirm the effectiveness as refrigerants of the combustion gases. From this point of view steam injection reduces NO emission up to 16% (in terms of mass fraction) when a steam flow which doubles the fuel flow is introduced, whereas reduction of about 8% is found using water injection in the same proportion to the fuel flow. Krishna and Ganesan [10] had done CFD analysis of flow through vane swirlers. The present work reports a computational study of steady flow through vane swirlers for various vane angles from 15° to 60° in steps of 15°. Flow has been simulated by solving the appropriate governing equations, namely, conservation of mass and momentum using the SIMPLE-algorithm. Turbulence has been modeled using Reynolds's Stress Model (RSM). They found that the RSM seems to be appropriate for swirling flows, especially for flows with high swirl, as in the case of gas turbine combustors.

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 $k-\epsilon$ model for turbulent flow, PDF flame let model and eddy dissipation combustion models for non premixed gas combustion, and P-1 radiation model. Comparison of the PDF flame let and eddy dissipation combustion model for non premixed gas combustion is also presented in this study. The outcome of the work will help in finding out the geometry of the combustion chamber which will lead to less emission.

2 MODELING, MESHING AND BOUNDARY CONDITIONS

2.1 PDF Flame let Model

The mathematical equations describing the fuel combustion are based on the equations of conservation of mass, momentum, and energy together with other supplementary equations for the turbulence and combustion. The standard $k-\epsilon$ turbulence model is used in this study. The equations for the turbulent kinetic energy k and the dissipation rate of the turbulent kinetic energy ϵ are solved. For non premixed combustion modeling, the mixture fraction/PDF model is used. In non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. The PDF/mixture fraction model is used for non premixed combustion modeling. In this approach individual species transport equations are not solved. Instead, equation for the conserved scalar (f) is solved, and individual component concentrations are derived from the predicted mixture fraction distribution. The P-1 radiation model is used in this study to simulate the radiation from the flame.

2.2 Eddy Dissipation Combustion Model

In this study Eddy Dissipation Combustion Model is also used to understand the combustion process and comparison with the PDF flame let combustion model. The eddy dissipation model is based on the concept that chemical reaction is fast relative to transport processes in the flow. When reactants mix at the molecular level, they instantaneously form products. The model assumes that the reaction rate may be related directly to the time required to mix reactants at the molecular level. In turbulent flows, this mixing time is dominated by the eddy properties, and therefore, the rate is proportional to a mixing time defined by the turbulent kinetic energy, k and dissipation, ϵ . This concept of reaction control is applicable in many industrial combustion problems where reaction rates are fast compared to reactant mixing rates.

2.3 Geometry

The basic geometry of the gas turbine can combustor is shown in Fig. 1. The size of the combustor is 590mm in the Z direction, 220mm in the Y direction, and 250mm in the X direction. The primary inlet air is guided by vanes to give the air a swirling velocity component. The injection diameter of primary air injector diameter is 100 mm. The fuel is injected through six fuel inlets in the swirling primary air flow. The fuel injector diameter is 4.2 mm for one hole. The secondary air is injected in the combustion chamber through six side air inlets each with an area of 33.50 mm². The can combustor outlet has a rectangular shape with an area of 0.0150 m². The dimensions of this geometry are taken from the Chauki Ghenai [1].

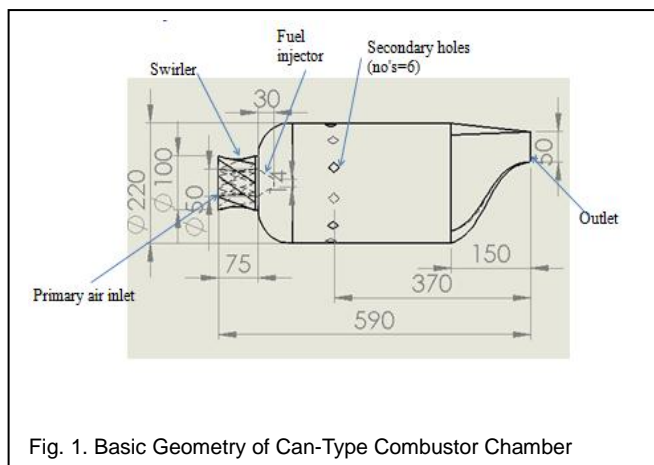


Fig. 1. Basic Geometry of Can-Type Combustor Chamber

2.4 Meshing

For the analysis of the combustion chamber, the commercial code CFX has been used in order to predict the centerline and the wall temperature distribution as well as combustion phenomena. Mesh generation is very important part of the work that has to be done before starting any CFD calculations. For this investigation, the grid generation has been done on Workbench – Mesh (ICEM CFD). The mesh is generated by automatic method is used for meshing then generate the mesh. The mesh consist of 1, 80,002 elements or cells and 34724 nodes and element type is tetrahedral.

2.5 Boundary Conditions in CFX Pre

The different boundary conditions applied for flow analysis of gas turbine can-type combustion chamber are taken from the Ghenai [1]. The boundary conditions of the primary air are: The injection velocity is 10 m/s, the temperature is 300 K, the turbulence intensity is 10%, mixture fraction $f = 0$. The boundary conditions of the fuel are: Mass flow rate, 0.001 Kg/s, the temperature is 300 K, the turbulence intensity is 10%, mixture fraction $f = 1$. The boundary conditions of the secondary air are : The injection velocity is 6 m/s, the temperature is 300 K, the turbulence intensity is 10%, mixture fraction $f = 0$ and the secondary air is injected in the combustion chamber through six side air inlets. The boundary condition of the outlet of combustion is defined by providing pressure value. The relative pressure is taken zero Pascal. The finite volume method and the first-order upwind method are used to solve the governing equations. The convergence criteria are set to 10⁻⁶ for the continuity, momentum, and turbulent kinetic energy, dissipation rate of the turbulent kinetic energy, energy and the radiation equations and the mixture fraction.

3 RESULTS AND DISCUSSION

3.1 Effect of Swirler Angle

For boundary conditions discussed in article 2.5 velocity contours are obtained at different swirler angle. Fig. 2. shows the contours of streamline from primary air inlet to outlet of combustor for swirler angle at 30°. Velocity at the entrance is more as compared to middle zone and finally at the exit it consolidates.

As we increase the angle of the swirler as 45° and 60° respectively the zone covered by the primary air at the inlet of combustion chamber is more. Also a recirculation zone is created after the swirler who helps in efficient combustion. Fig. 3. and Fig. 4. shows velocities streamline in can-type combustor when the swirler angle is kept 45° and 60° respectively. It is observed that at radial distance $r=R$, lower velocity is observed and from outer radius to center distance this velocity increases. Also it is observed that for lower swirl angle the highest velocity achieved is higher, so the residence time for the mixture is less. As this swirler angle increase the velocity decreases, so residence time for the mixture increases. This will in turn burn the fuel efficiently and reduce the pollutants emissions. Due to proper mixing of fuel and air it will enhance the combustion phenomena leading to less emission.

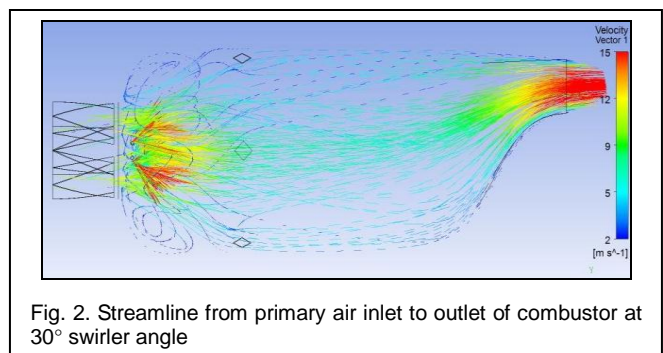


Fig. 2. Streamline from primary air inlet to outlet of combustor at 30° swirler angle

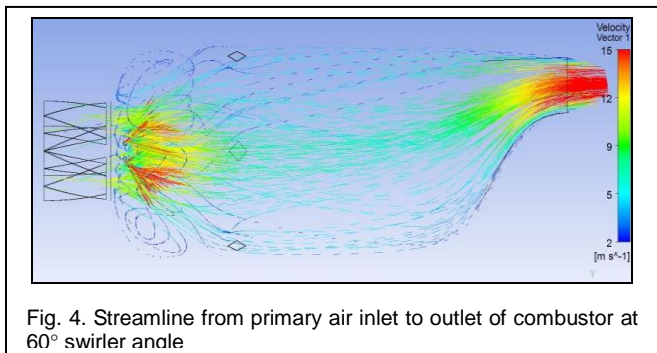
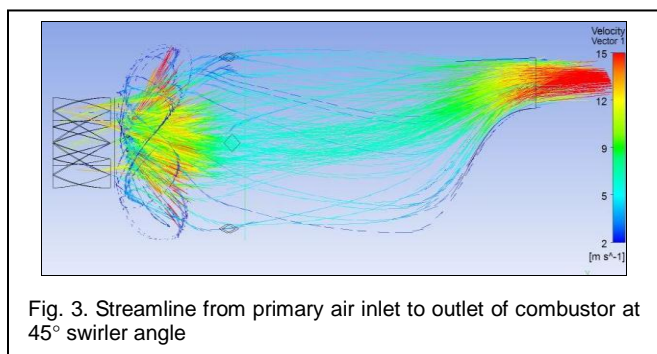


Fig.5 shows the profile of temperature along center line. From this chart it is observed that the exit temperature achieved for 30° swirler is maximum compared to the 45° and 60°. At 60° lowest exits temperature is achieved. The NO_x emission is directly concerned with the temperature. The more the exit temperature more will be the NO emission. So the case of 60° swirler angle geometry of combustion chamber is best compared to the other swirler angle cases.

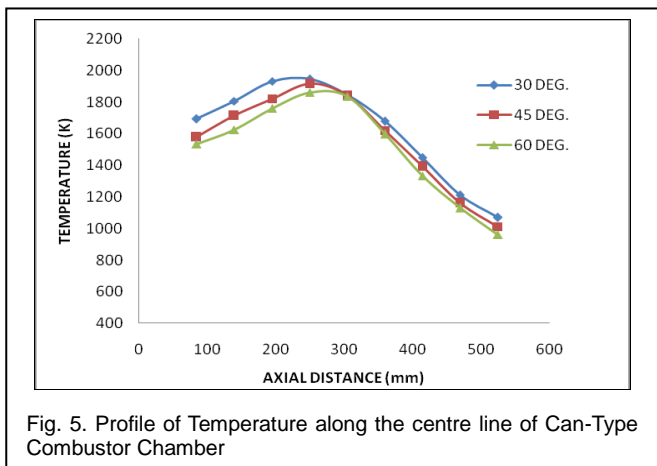
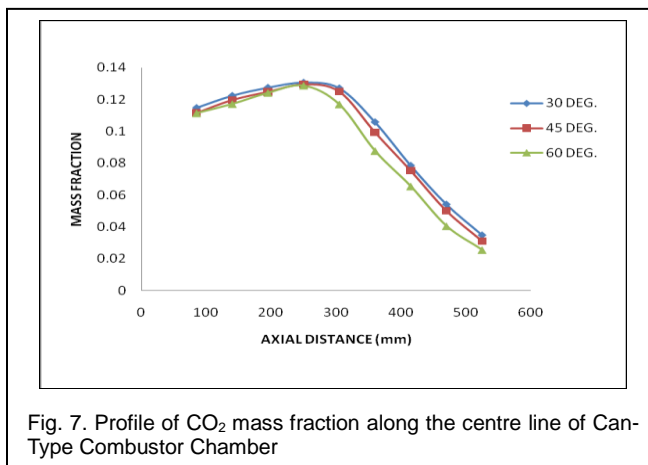
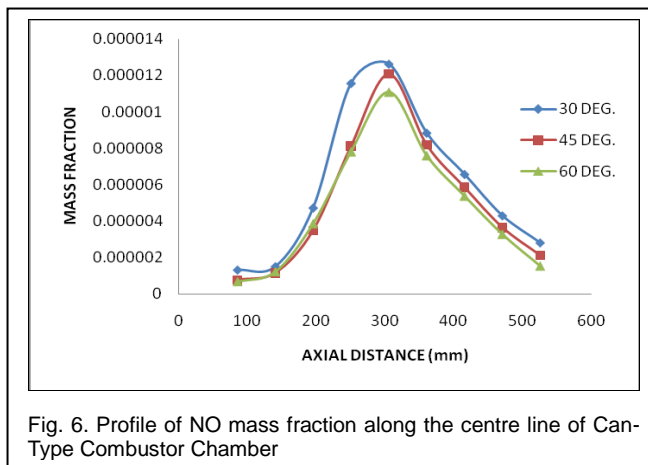


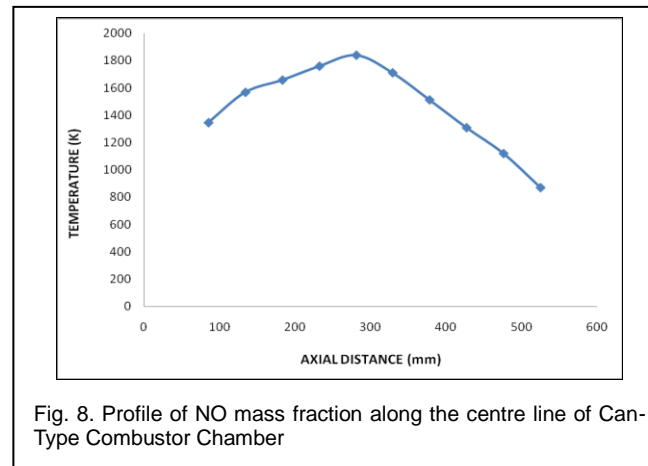
Fig. 6. depicts the profile of NO mass fraction along center line of the combustion chamber. From this profile it is observed that in 60° the lowest NO is found. From this we can conclude that NO is directly concerned with the temperature. It is thus important to reduce the temperature of gases coming out of the combustion chamber. This is achieved by mixing of air is sufficient quantity at secondary air inlet in combustion chamber. Fig. 7. shows the profile of CO₂ mass fraction along center line of the combustion chamber. It is observed from the profile that less mass fraction of CO₂ is achieved at 60° swirler geome-

try. In the case of 30° and 45° swirler the exit mass fraction of CO₂ are nearly same.



3.2 Parametric study

The contour of the predicted gas temperature for the combustion of methane in gas turbine can-combustor is shown in Fig.8. The maximum gas temperature for methane combustion is 1850 K. For the validation of the combustion model, the predicted flame temperature for methane combustion is compared to the adiabatic flame temperature. For natural gas or methane fuel and with initial atmospheric conditions (1 bar and 200 C), the theoretical flame temperature produced by the flame with a fast combustion reaction is 1950 K.



The predicted maximum temperature of the combustion products or the adiabatic flame temperature compares well with the theoretical adiabatic flame temperature. Fig. 8. shows the profile of temperature along center line. The peak gas temperature is located in the primary reaction zone where the combustion of mixture of air and methane takes place. The fuel from the six injectors is first mixed in the swirling air before burning in the primary reaction zone. The gas temperature decreases after the primary reaction zone due to the dilution of the flame with the secondary air.

Temperature contours are plotted at different axial position to understand the propagation of combustion flame in chamber. Fig. 9. (a),(b),(c), and (d) shows the temperature contours (X-Y Plane) for methane combustion in can-combustor at axial positions of $Z = 100$ mm, 200 mm, 300 mm, and 400 mm respectively. The first contour ($Z= 100$ mm) represents the gas temperature near the six fuel injections. The fuel is injected from the six fuel inlets and mixed with the swirling air before the start of the combustion. The size of the flame increases downstream and reaches a maximum at radius $Z = 200$ mm. Clear flame visualization is observed having maximum temperature of 2000 K. The radius and temperature of the flame decrease after that with the increase of the axial distance ($Z= 300$, and 400 mm).

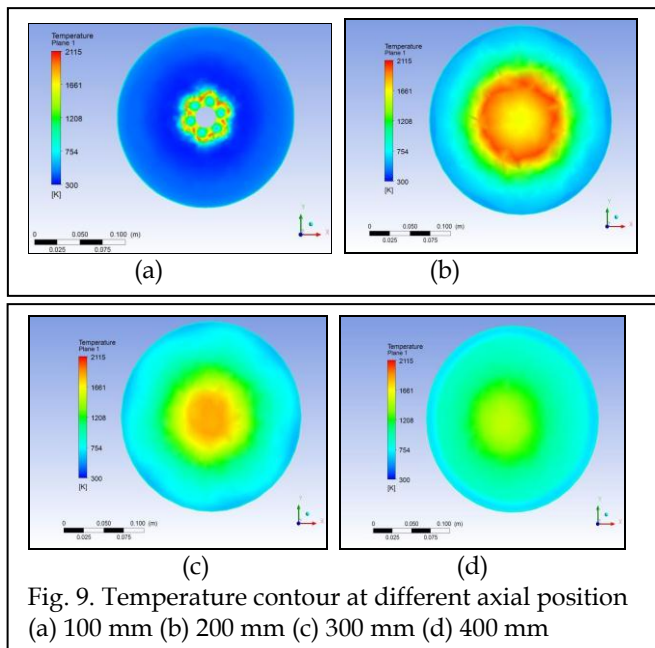
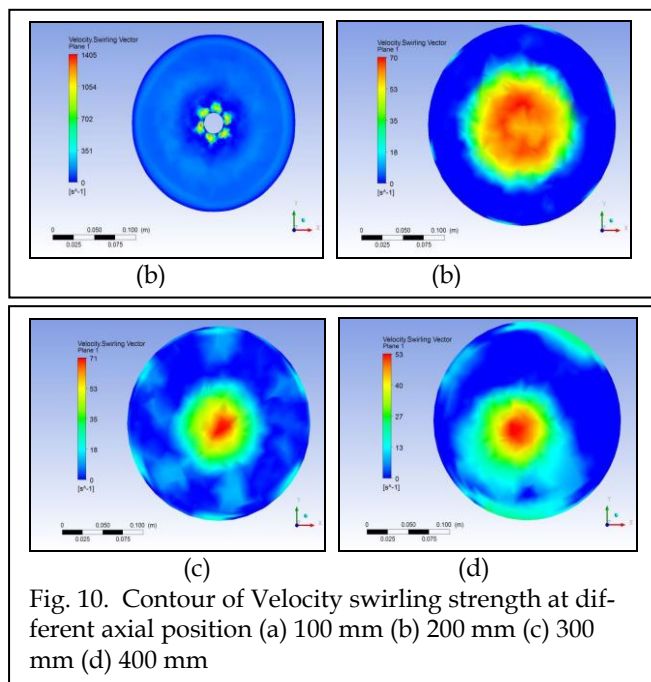


Fig.10 (a), (b), (c), and (d) shows the contours (X-Y Plane) of the velocity swirling strength for $Z = 100$ mm, 200 mm, 300 mm, and 400 mm respectively. Strong recirculation regions with strong swirling strength are observed at the entrance regions near the fuel injection. The contours show that the velocity swirling strength decreases downstream with the increase of the axial distance Z . The velocity swirling strength contours at $Z = 200$ mm shows an annulus region with high swirling strength near the secondary air inlet position. At $Z = 100$ mm there is much variation in positive and negative axial velocity- w , it means at wall surfaces of combustor the velocity is nearly zero, it becomes positive towards center and again it

deviates after getting peak reaching zero at other surface. This tremendous variation of velocity means creation of strong recirculation zone which leads the good mixing of air and fuel for complete combustion. At $Z = 300$ mm and $Z = 400$ mm deviation of velocity becomes less and magnitude of velocity is only positive and are not capable to produce strong recirculation zone which help in moving gases out of the combustion chamber.



In this case the study is carried out by changing the equivalence ratio. Equivalence ratio is the ratio of actual fuel-air ratio to Stoichiometric fuel-air ratio. Different equivalence ratio is taken to see the effect of it on temperature and emission level. For lesser equivalence ratio, the maximum achieved temperature is less and flame length is short. As the equivalence ratio increases both maximum temperature and flame length increases significantly. This is due to the fact that increment in equivalence ratio means more mass flow rate of fuel is consumed.

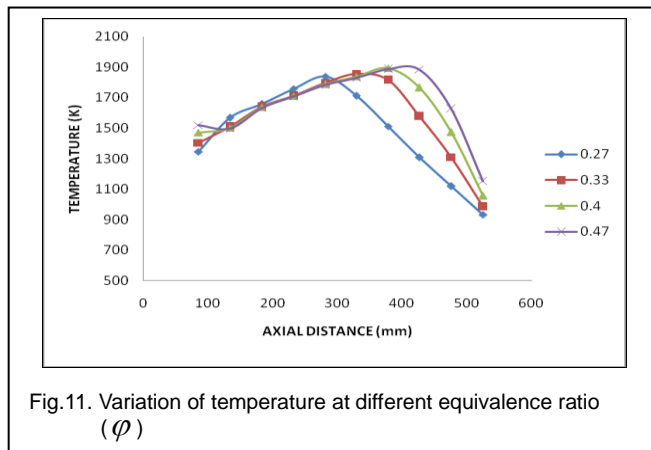


Fig. 11. shows the variation of temperature at different equivalence ratio (ϕ). In this profile highest temperature achieved at

different location and temperature variation is also different. It is observed that as equivalence ratio increases the variation in temperature increases. Fig. 12. shows the variation of NO mass fraction at different equivalence ratio (ϕ). In this chart we can see that at lowest equivalence ratio $\phi = 0.27$, the value of achieved NO mass fraction is lowest. For equivalence ratio $\phi=0.33, \phi=0.40, \phi=0.47$, the NO mass fraction increases respectively. The highest value of NO mass fraction is achieved for $\phi=0.47$ because as equivalence ratio increases more fuel is burned in combustion.

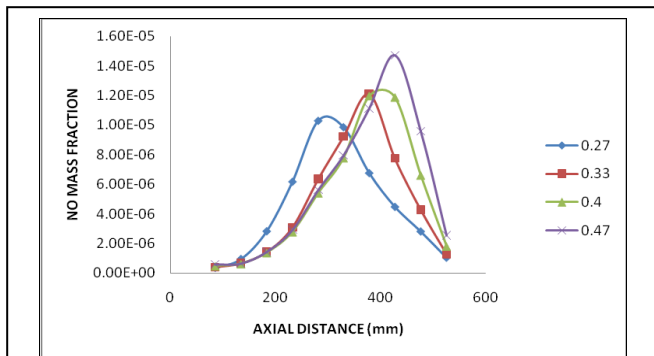


Fig.12. Variation of NO mass fraction at different equivalence ratio (ϕ)

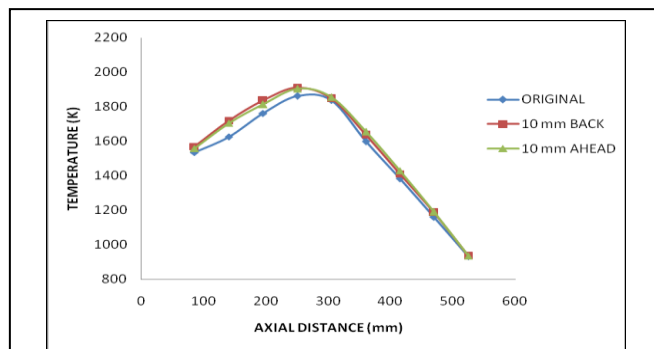


Fig.13. Effect of shifting dilution hole on variation of temperature

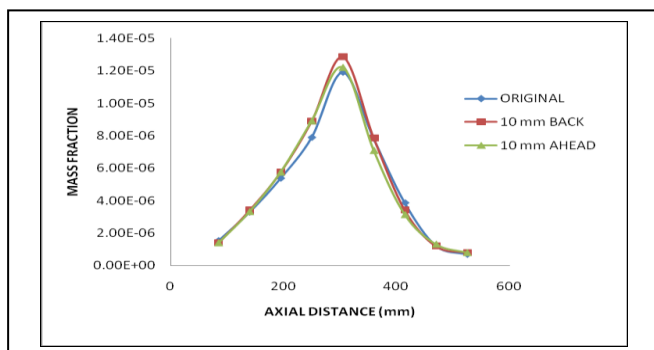


Fig.14. Effect of shifting dilution hole on NO mass fraction

The position of dilution hole is changed to see the variation on temperature and on NO mass fraction. The position of dilution holes are shifted 10 mm ahead and 10 mm back than the original geometry. The effect can be observed from the charts. Fig. 13. shows the effect of shifting dilution hole on variation of temperature. Due to shifting of dilution hole, it is observed

that at reaction zone there is variation in temperature. Temperature for 10 mm back position of dilution holes is having the highest temperature. Temperature for 20 mm back position of dilution holes is also having the higher temperature than the original geometry. But after further movement along the axial direction the variation of temperature is not much more. Fig. 14. shows the effect of shifting dilution hole on NO mass fraction. It is quite observed from this profile that at axial distance $Z=250$ mm, mass fraction of NO is higher in case of 10 mm back position of dilution holes where temperature is also higher. Mass fraction of NO for 20 mm back position of dilution holes is also having the higher mass fraction of NO than the original geometry. But after further movement along the axial direction the variation of mass fraction of NO is not much more.

4 CONCLUSION

Numerical investigation on Can-type combustion chamber shows following findings:

- 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.
- For methane as fuel and with initial atmospheric conditions, the theoretical flame temperature produced by the flame with a fast combustion reaction is 1950 K. The predicted maximum flame temperature is 1850 K of the combustion products compares well with the theoretical adiabatic flame temperature.
- Temperature profiles shows increment at reaction zone due to burning of air-methane mixture and decrement in temperature downstream of dilution holes because more and more air will enter in combustion chamber to dilute the combustion mixture along center line. Specie namely NO is increasing and achieving peak point at reaction zone because they are products of combustion along center line.
- Due to increase in equivalence ratio, temperature and mass fraction of NO increases because more fuel is utilized. There is not much variation in temperature and NO emission by shifting the axial location of dilution holes.

REFERENCES

- [1] Chaouki Ghenai, "Combustion of Syngas Fuel in Gas Turbine Can Combustor", *Hindawi Publishing Corporation, Advances in Mechanical Engineering*, vol. 2010, 13 pages, Article ID 342357, doi:10.1155/2010/342357
- [2] J. J. McGuirk, and J.M.L.M Palms, "Experimental Investigation of the flow inside a water Model of a Gas Turbine combustor. Part 1 - Mean and Turbulent Flow field." *J. Fluids Eng.*, vol. 117, pp. 450 - 458, Sep. 1995.
- [3] M. N. Mohd Jaafar, K. Jusoff, Mohamed Seroleh Osman, and Mohd Shaiful Ashrul Ishak, "Combustor aerodynamic using radial swirler", *Int. J. of the Physical Sciences*, vol. 6(13), pp. 3091 - 3098, Jul. 2011.
- [4] P. Koutmos and J.J. McGuirk, "Investigation of Swirler/Dilution Jet Flow Split on Primary Zone Flow patterns in a Water Model Can-Type Combustor", *J. Engineering for Gas Turbine and Power*, vol. 111, pp. 310 - 317, Apr. 1989.

- [5] K.Y.Sokolov, A.G.Tumanovskiy, and M.N.Gutnic, A.V.Sudarev, Y.I.Zakharov, and E.D.Winogradov, "Mathematical Modeling of an Annular Gas Turbine Combustor", *J.Engineerig for Gas Turbine and Power*, vol. 117, pp. 94 - 99, Jan. 1995.
- [6] Kulshreshtha D B, Channiwala S. A., Chaudhari K. V and Dikshit S. B. , "Design, Numerical Simulation and Experimental Investigations on Can Annular Type Combustion Chamber for 20 kW Gas Turbine Engine." *Int. J. Dynamics and Fluids*, vol. 4, no. 2, pp 175 - 189, 2008.
- [7] H. J. Tomczak, G. Benelli, L Carrai, and D Cecchini, "Investigation of a Gas Turbine combustion system fired with mixtures of Natural Gas and Hydrogen." *IFRF Combustion Journal*, Dec. 2002, 19 pages, Article ID 200207.
- [8] Tomohiko Furuhata, Shunsuke Amano, Kousaku Yotoriyama, and Masataka Arai, "Development of can-type low NOx combustor for micro gas turbine", *J. Fuel*, vol. 86, pp 2463 - 2474, 2007.
- [9] Ernesto Benini, Sergio Pandolfo, and Serena Zoppellari, " Reduction of NO emission in turbojet combustor by direct water/steam injection: Numerical and Experimental assessment", *J. Applied Thermal Engineering*, vol. 29, pp 3506 - 3510, 2009.
- [10] S Bharat Krishna, and V. Ganesan, "CFD analysis of flow through Vane Swirlers", *J. Institution of Engineers*, vol. 86, pp 12 - 19, 2005.