Numerical Analysis of Combustion Chamber with Methane Air Mixture for Different Inlet Air Velocities

Priyank Shah, Samir Mehta, Rushikesh Gaikwad, Nitin Pandey

Abstract— Three dimensional numerical investigation of Can type combustor is presented in this study. This study uses CFX solver for the investigation. The objective is to study the combustion phenomenon and the effects of varying inlet velocity of air on the combustion taking place inside the combustor. Before any prototype is built for testing, it is always tested on CFD for the design conditions, operating conditions and the transient conditions. Through this study major effects on the design can be simulated quite efficiently and hence it can be analysed. This causes reduced number of revisions in the design and more compact structure. Methane-Air mixture is used as Combustion Mixture. The can type combustor is basically used for gas phase fuels like methane and propane. Reacting mixture is not premixed in this study. Parameters like inlet air velocity, swirler angle at the primary air inlet, air-fuel ratio have been varied in order to better understand the combustion phenomenon. Eddy dissipation combustion model is primarily used in this study. This study will help in finding out the range of inlet air velocity at which the predefined combustor model will be most efficient.

Index Terms— Combustion Analysis, Can type, Combustor, CFD, CFX, Methane Air mixture,Inlet air velocity, Eddy dissipation combustion model.

◆

1 INTRODUCTION

The combustion process increases the internal energy of a gas, which translates into an increase in temperature, pressure or volume depending on the configuration. In an enclosure, for example the cylinder of a reciprocating engine, the volume is controlled and the combustion creates an increase in pressure. In a continuous flow system, for example a jet engine combustor, the pressure is controlled and the combustion creates an increase in volume. This increase in pressure or volume can be used to do work, for example, to move a piston on a crankshaft or a turbine in a gas turbine. If the gas velocity changes thrust is produced, such as Exit nozzle of a rocket engine.

The combustion chamber in gas turbines and jet engines (including ramjets and scramjets) is called the combustor. It is also known as a burner, combustion chamber or flame holder because it hold the flame into it. In a gas turbine engine, the combustor or combustion chamber is fed high pressure air by the compression system. The combustor then heats this air at constant pressure. After heating, air passes from the combustor through the nozzle guide vanes to the turbine. In the case of a ramjet or scramjet engines, the air is directly fed to the nozzle. Generally open cycle gas turbines use air fuel ratio in the range of 50:1 to 200:1. Mixture has to be within flammability limits. The flame (combustion) must be held inside of the combustor. If combustion occurs further back in the engine, the turbine stages can easily be damaged. Additionally, with advancement turbine blades are able to withstand higher temperatures, the combustors are being designed to burn at higher temperatures and the parts of the combustor need to be designed to withstand those higher temperatures. Various different types of fuels can be used but mostly natural gas and its derivatives are used. Fuel used in this study is Methane as this was the Output Proposed by the Anaerobic Digestor.

- *Pathan, Patel and Tadvi [1]* carried out numerical investigation of process of combustion of Methane air mixture in a can type combustor. They analysed the effect of shifting dilution holes and changing swirler angle in the combustor on emissions due to combustion of fuel. The results showed that 60°swirler geometry is giving less NO emissions the temperature at the exit of combustion chamber is less as compared to 30°and 45°swirler angle geometry.
- *Chaouki Ghenai [2]* studied the impact of the variability in the alternative fuel composition and heating value on combustion performance and emissions. The gas turbine can combustor is designed to burn the fuel efficiently, reduce the emissions, and lower the wall temperature. Syngas mixtures with different fuel compositions are produced through different coal and biomass gasification process technologies. 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. The mathematical models used for syngas fuel combustion consist of the k-ε model for turbulent flow, mixture fractions/PDF model for nonpremixed gas combustion, and P-1 radiation model.
- *Cyrus B. Meher-Homji et. al. [3]* have done extensive research on gas, liquid and alternative new fuels for gas turbines and interrelationships of fuel system design, fuel properties and gas turbine operability in terms of dry low NOx/dry low emissions(DLN/DLE) combustion. It covers mechanical drive gas turbines, and smaller engines commonly used in oil and gas markets, and large advanced gas turbines used in power generation and combined cycle applications. Case studies related to failures due to fuel related

problems are provided to indicate the importance of fuel treatment and quality control. It compares fuels like Methane and Propane.

- *Jaafar et. al.[4]* investigated flows for different swirler angles in a gas turbine combustion chamber . They found that highest swirl number for flat vane was 2.29 and for curve vane was 1.57. It also showed that higher the swirl strength lesser is the corner recirculation zone. Parametric study carried out showed for producing appropriate recirculation zone swirler angle of about 50° was best.

In this study parameters like Inlet air velocity, swirler angle and Air-Fuel ratio have been varied to choose the most Optimal Values for achieving near Complete Combustion. Effects of changing these parameters on the combustion inside the chamber have been studied. Software used for modeling is SolidWorks'13 and analysis is being done on Ansys CFX14.0 Models suitably used in studying the combustion phenomenon are k-ε model for turbulent flow, eddy dissipation combustion model for non premixed gas combustion and P-1 radiation model. The Toroidal recirculation was formed which resulted in High levels of Turbulence Energy in the core of combustor at low level of swirler flow rate. The outcome of this study will help in understanding combustion phenomenon inside the combustion chamber according to the varying parameters and also it will help in determining the optimum inlet air velocity for a particular swirler angle. The exit temperature of the combustor was tried to keep well under the Melting temperature, thus reducing the thermal fatigue cycle on the body of combustor. As this Outlet gases were studied to be transferred to the Turbocharger Turbine Stage, the blades of the turbine has a limit on the inlet temperature of the Gases, above which they can lower the efficiency.

2 MODELLING OF GEOMETRY

Below is the Snap of the modeled Combustor Chamber 2D Drafted View. The dimensions of the combustion chamber have been taken from Chaouki Ghenai [2]. The Cross Sectional Area of the Air Inlet, Gas Inlet and Gas Mixture Outlet was kept same. Methane have a Lower Explosive Limit(LEL) of 5% and Upper Explosive limit(UEL) of 15% and any concentration above or below this will lead the Combustion to Extinguish, thus the provision was made for the inlet of the Secondary Air in addition to Primary air inlet along with the fuel. The Fuel Inlet slots are placed at an angle and made symmetric so that the flame can envelope whole Combustor uniformly. Outlet of the Combustor was designed to converge to match the size with the Inlet of the Turbocharger Inlet. The curvature was achieved by keeping the Tangency condition at the Periphery of the Combustor and perpendicular to the Outlet. Combustor dimensions are shown below:

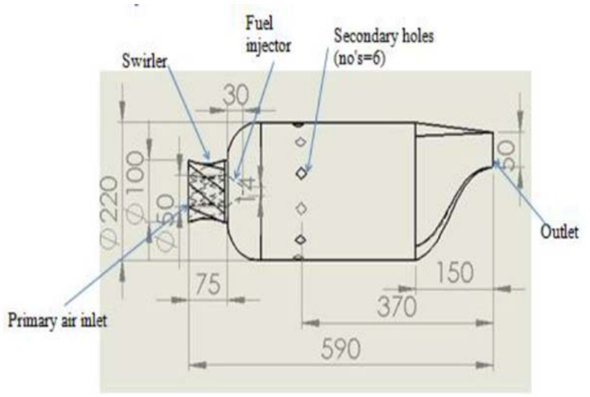

Fig. 1. Geometry of Can-Type Combustion Chamber.

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. Primary air is introduced in the chamber through an annular area with outer diameter of 100mm and inner diameter of 50mm. Inlet of fuel ,which is Methane in this study, is through six holes in the swirling primary air flow each with a diameter of 4.2mm. It has six secondary holes for air inlet each of area 33.50 mm². The outlet has an area of 0.0150 m² which are rectangular in shape.

3 MESHING OF GEOMETRY

Geometry modeled in SolidWorks is imported in Ansys Mechanical which was further linked to ANSYS CFX Pre and ANSYS CFX Results. The mesh type used in this study is Tetrahedral. Mesh size selected is medium and Mesh size was kept at default. The number of nodes is 32883 and elements are 175779.

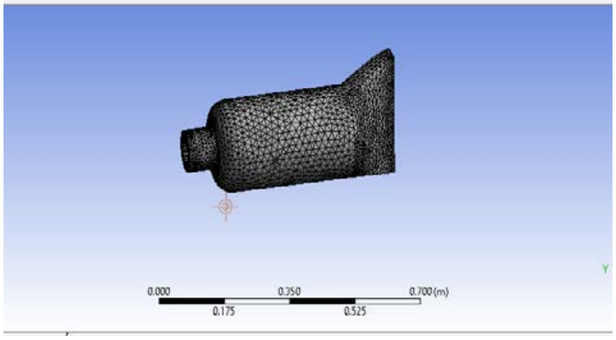

Fig. 2. Meshed Geometry of Combustion Chamber in Ansys Workbench

---

- *Author Priyank Shah is currently in process to apply for masters degree program in mechatronics engineering ,PH-0918879086557. E-mail: shahpriyank9@yahoo.in*
- *Author Samir Mehta is currently working as Associate Consultant in Fintellix Solutions Pvt. Ltd. , PH-0919930213918. E-mail: samir201243@yahoo.com*
- *Author Rushikesh Gaikwad is currently working as Trainee engineer in Infosys ,PH-0917387730666 , E-mail : rgaikwad610@gmail.com*
- *Author Nitin Pandey is currently working as Trainee Engineer –Vessels in Aker Solutions Pvt. Ltd., PH-0918898885987, E-mail : nitinpandey2911@gmail.com*

4 EDDY DISSIPATION MODEL

Eddy Dissipation Method was used for solving the problem. The Eddy Dissipation model tracks each individual chemical species (except for the constraint material) with its own transport equation. EDM Coefficient value was taken as 0.5, this value was referenced from a similar Sum in the Ansys solved problem. This model is flexible so that one can readily add new materials, such as additional fuels, tuning existing fuel composition to suit one's simulation requirement without complications. A limitation of this model is that radical or intermediate species, such as CO, cannot be calculated with adequate accuracy. This may lead to over-prediction of flame temperature, in particular in fuel-rich regions.

5 BOUNDARY CONDITIONS

Boundary conditions were set in CFX-Pre. Initial Boundary conditions were taken from paper written by *Pathan, Patel and Tadvi* [1]. The initial conditions taken are as follows:

Primary Air: Velocity: 10m/s, Temperature: 300K, Mixture fraction f=0.

Secondary Air: Velocity: 6m/s, Temperature: 300K, Mixture fraction f=0.

Fuel: Mass flow rate: 0.001kg/s, Temperature: 300K, Mixture fraction f=1.

Turbulence intensity: 10%.

Outlet Conditions: Considered in terms of Pressure. Relative pressure is taken as 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 maintaining the continuity, momentum and turbulent kinetic energy, energy and the radiation equations and the mixture fraction. The finite volume method and the first-order upwind method are used to solve the governing equations.

6 RESULTS AND DISCUSSIONS

6.1 Temperature profile for Eddy dissipation method

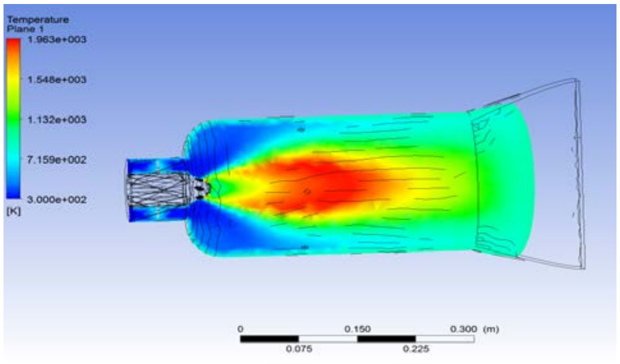

Fig. 3. The temperature profile for the combustion chamber for Swirler angle of 60 ° with initial inlet and outlet conditions

Above picture shows the temperature profile for the combustion chamber for Swirler angle of 60 ° with initial inlet and outlet conditions. The image shows the process of combustion inside the chamber. The red part denotes the actual temperature of the zone where the combustion happens. This temperature is around 1950 K. The blue part denotes the temperature surrounding the zone which gradually decreases as we move towards the wall of the chamber. Thus this temperature variation shows the combustion is contained within the combustion chamber.

The parameters that were varied during this study are as follows:

1. Inlet air velocity – varied from 6m/s to 20m/s.
2. Equivalence ratio– 0.573, 0.43, 0.344, 0.2866.
3. Swirler angles – 30°, 45°, 60°.

6.2 Streamline flow in the domain for different Swirler Angles

The Swirler Angle was varied in the CAD model which was rerouted to the Mesh thus keeping the other parameters of the setup same. Angle was attained by given a twist value in the model from 30^0, 45^0 & 60^0. Below shown are some streamlines plotted from primary inlet to the outlet using Streamlines Function, the streamlines show the path what the fluid inside the combustor followed along the length of the combustor. View was changed to wireframe type to clip the streamlines clearly.

For 30° Swirler Angle:

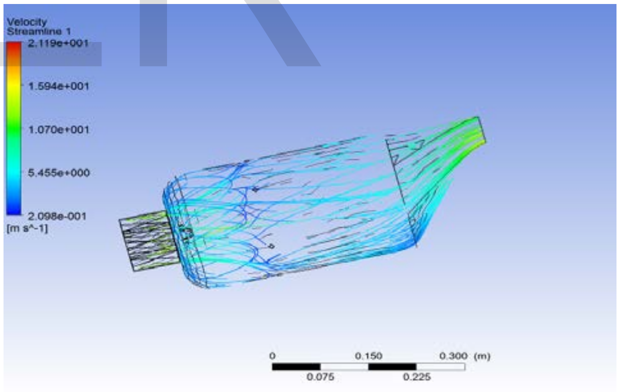

Fig. 4. Streamline Flow for 30° Swirler Angle

For 45° Swirler Angle:

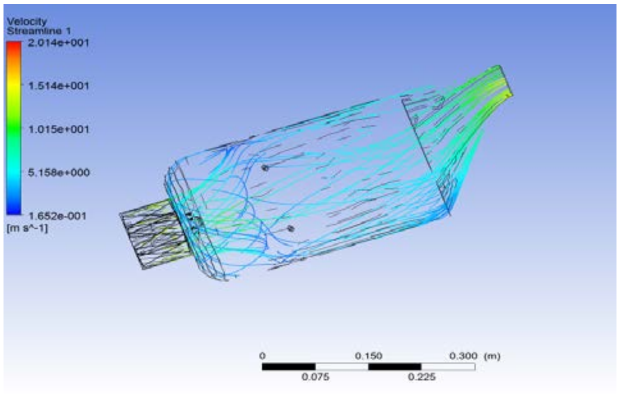

Fig. 5. Streamline Flow for 45° Swirler Angle

For 60° Swirler Angle:

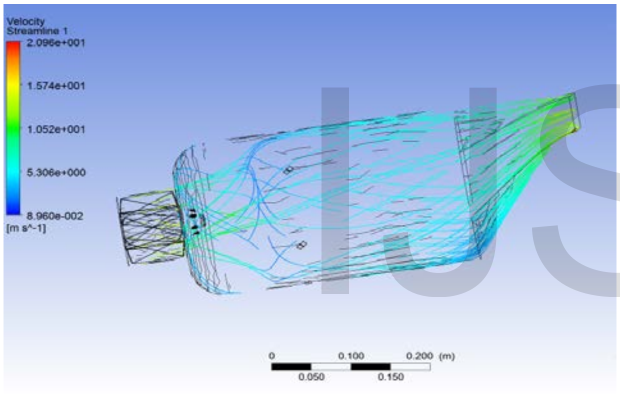

Fig. 6. Streamline Flow for 60° Swirler Angle

Velocity at the entrance is more as compared to middle zone and finally at the exit it consolidates. As we increase the angle of 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 swirler which helps in efficient combustion. 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 low swirler angle the maximum velocity achieved is higher, so the residence time for the mixture is less. As this swirler angle increases the velocity decreases, so residence time for the mixture increases. This will in turn burn the fuel efficiently and reduce the pollutant emissions. Due to proper mixing of fuel and air it will enhance the combustion phenomena leading to less emission.

6.3 Plot of maximum outlet temperature versus inlet air velocity for 60° swirler angle

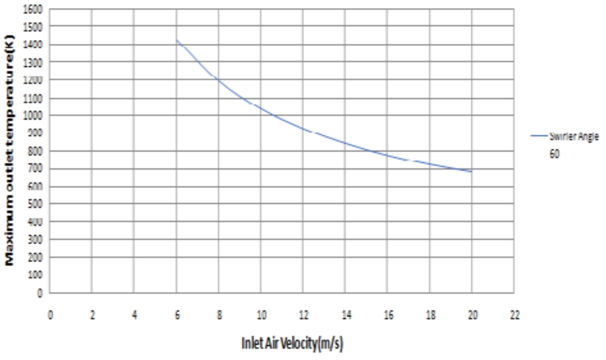

Fig. 7. Plot of maximum outlet temperature versus inlet air velocity for 60° swirler angle

The plot shows the variation of Maximum outlet Temperature with different inlet air velocities for 60°swirler angle. As the concentration of NOx emission depends on the outlet temperature, lower the outlet temperature lower is the concentration of NO in exhaust. Lower outlet temperature is preferred so as to avoid damage to the turbine blades if the exhaust is used for power generation.

6.4 Plot of Maximum Exhaust Velocity versus Inlet Air Velocity for different Swirler Angles

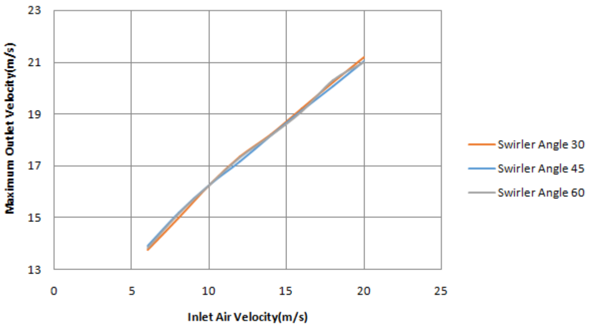

Fig. 8. Plot of Maximum Exhaust Velocity versus Inlet Air Velocity for different Swirler Angles

The plot shows the variation of maximum outlet velocity versus different inlet air velocities for three different swirler angles. It is seen that the outlet velocity gradually increases as inlet velocity is increased. It can be noted that the curve follows almost a straight line. It is also observed that for a small increase in outlet velocity the amount by which the inlet velocity should be increased is more. The maximum inlet velocity which could be increased for successful convergence of the profile is 20m/s provided all the other boundary conditions remain the same.

6.5 Plot of Maximum Temperature obtained in the combustion chamber versus Inlet Air Velocity for different Swirler Angles.

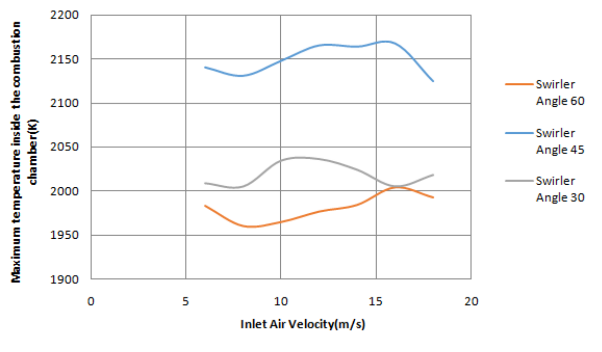

Fig. 9. Plot of Maximum Temperature obtained in the combustion chamber versus Inlet Air Velocity for different Swirler Angles.

The plot shows variation of the maximum temperature attained in the combustion chamber versus inlet air velocity for three different swirler angles. It can be seen that the temperatures attained initially increase for all the swirler angles and then decline after a certain velocity. The velocity after which the temperature starts dropping is different for different swirler angles. It is also observed that average maximum temperature is more for swirler angle of 45° followed by 30° and then 60°. It can be concluded that more proper mixing is done in 45° swirler angle.

6.6 Plot for Temperature versus Axial distance for different Swirler Angles

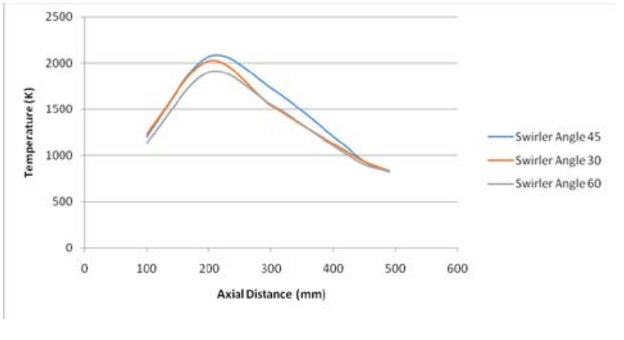

Fig. 10. Plot for Temperature versus Axial distance for different Swirler Angles.

The above plot shows variation of temperature along the axial length of the combustion chamber for three different swirler angles.The plot shows the temperature gradually increases as we travel along the chamber attains a maximum value then decreases as we reach towards the outlet .Axial distances where temperature reaches its peak is different for different swirler angles. It is observed that maximum temperature is attained at an earlier period for swirler angle 30°.Then followed by 60° then 45°. It can also be observed that high temperature exists for a greater part of the combustion chamber for swirler angle 45° then 60° and 30°.

6.7 Plot for Temperature versus Axial distance for different Equivalence Ratios

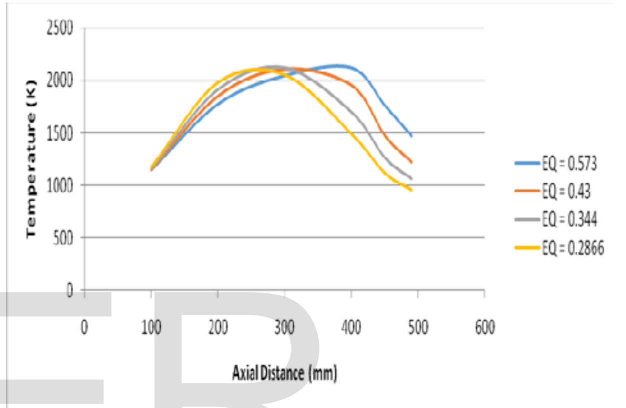

Fig. 11. Plot for Temperature versus Axial distance for different Equivalence Ratios.

The plot shows variation of temperature along the axis of the chamber for different equivalence ratios. Equivalence ratio is the ratio of actual fuel-air ratio to Stoichiometric fuel-air ratio *[1]*. The plot is done for 45° swirler angle for test velocity of 10m/s. It is observed that temperature peaks are attained at different axial distances for different equivalence ratios. It is seen that as the ratio increases the distance at which temperature peak is obtained shifts towards right thus increasing the wall temperature and the outlet temperature.

8 CONCLUSION

The following can be concluded from the results:

1. Maximum temperature in combustor is more for swirler angle 45°.Hence it can be concluded that more proper mixing is done in 45° swirler angle.
2. Lower the equivalence ratio lower is the wall and outlet temperature. Thus, more uniform combustion is achieved for lower equivalence ratios.
3. High temperature exists for a greater part of the combustion chamber for swirler angle 45° then 60° and 30°.

4. It is economical to operate the combustion chamber at an inlet velocity of range 10-14 m/s as the resulting increase in outlet velocity is comparatively high.
5. As per plot of maximum outlet temperature v/s inlet air velocity, outlet temperature decreases as the inlet air velocity increases and attains a constant value for further increase.

ACKNOWLEDGMENT

The authors wish to thank the Honorable Principal of K.J.Somaiya College of Engineering, Dr. Shubha Pandit who made all the facilities available for us in the college premises and Head of Department of Mechanical Engineering, Prof. Ramola Sinha for her untiring encouragement and guidance. It is indeed a matter of great pleasure & privilege to be able to present this synopsis report on “Numerical Analysis of Combustion Chamber with Methane Air Mixture for Different Inlet Air Velocities” under the valuable guidance and support of Prof. S.R. Nikam. We also thank librarian Ms. Deepali Kuberkar for the library facilities and access to technical papers provided which proved elemental to the success of the project. It has been great fun to work together and troubleshoot problems. Moral support from our parents is gratefully acknowledged.

REFERENCES

[1] 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 International Journal of Scientific & Engineering Research, Volume 3, Issue 10, October-2012 (ISSN 2229-5518).

[2] 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

[3] “Gas Turbine Fuels- System Design, Combustion and Operability” by Cyrus B. Meher-Homji, Justin Zachary, Andrew F. Bromley

[4] 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.