Analysis of Gas Turbine Combustor with Varying Air Inlet Velocity and Methane Fuel Inlet Temperature

K Guruvenu[1]* and B Konda Reddy[2]
[1]M. Tech research scholar, [2]Professor
Dept. of Mechanical Engineering, PBRVITS, Nellore, AP, India-524201

ABSTRACT

Numerical investigation of combustion effectiveness and properties of methane air mixture is tested in gas turbine can type combustor through three-dimensional computational fluid dynamics by using ANSYS CFX 14.5. In this study we designed two models of combustors. Using modified and basic model we compared the performance and efficiency of combustors, we seen approx. 23% reduction in temperature and 76% decrement in emissions. Using modified model methane air mixture is tested under normal and preheated conditions which helps to analyze the combustion properties and how the emissions (CO_2 & NO_x) will vary, we seen approx. 9% increment in temperature and 57% decrement in emissions.

KEYWORDS: Gas turbine combustor, ANSYS 14.5, Computational Fluid Dynamics

I. INTRODUCTION

Gas turbine engines are mostly used in the modern world to generate the power or thrust, they are called mostly eligible to generate thrust for aircrafts. The combustor is one of the parts of turbine engine, we have different stages in gas turbine engine. In this study we deal with combustion stage or combustion chamber. In this stage the power or energy is added to the engine in the form of heat and it helps to rotate the multistage turbines to generate power to run the compressor etc. the main aim of the combustor is to design it with low volume less emissions. Flow field in the combustion is the main part which has to be studied and analyzed both in academics and industries, it has the commercial importance to analyze and predict at various phenomena during combustion. Gas turbine combustor has to be developed and designed to meet many mutual conflicting design requirements i.e., High combustion efficiency over a wide operating envelope, Low NOx emissions, Low smoke, Low flame stability limits, Good starting properties, Low pressure drop and sufficient cooling air to maintain low wall temperature levels with structural durability. Numerical simulations on combustor flow fields had become unavoidable way to accelerate the design modifications which have the best efficiency and to optimize their performances. Numerical calculations over the software also facilitates the understanding and visualization of physical phenomena which costs high or often inaccessible by experimental way. The use of CFD codes to predict the flow fields with in a gas turbine combustor has been motivated for many studies. In this work three dimensional turbulent fluid flow analysis is carried out by using swirl stabilized gas turbine combustor model with compressible CFD computations were performed using ANSYS CFX employing k-ἐ turbulence model for closure and eddy dissipation model for combustion. In this work we analyzed the different properties over basic combustor design and modified combustor design with Methane-Air mixture at same input conditions and fuel preheated performance is analyzed for modified combustor to calculate or predict how NOx mass fraction will vary.

II. GEOMETRY

The basic geometry of the gas turbine can-type combustor chamber is shown in Fig. 1. The size of the combustor is 590 mm in the Z direction, 250 mm in the Y direction, and 230 mm in the X direction. The primary inlet air is guided by vanes to give the air a swirling velocity component. The total surface area of primary main air inlet is 57 cm^2. The fuel is injected through six fuel inlets in the swirling primary air flow. There are six small fuel inlets, each with a surface area of 0.14 cm^2. The secondary air is injected in the combustion chamber through six side air inlets each with an area of 2cm^2. The secondary air or dilution air is injected at 0.1 m from the fuel injector to control the flame temperature and NOx emissions. The can-type combustor outlet has a rectangular shape with an area of 0.0150 m^2. For modified two extra air inlet slits (vertical and horizontal) of 10 and 15mm near primary inlet and near nozzle inlet is added because of that we can reduce the combustor wall temperature levels and increase its life.

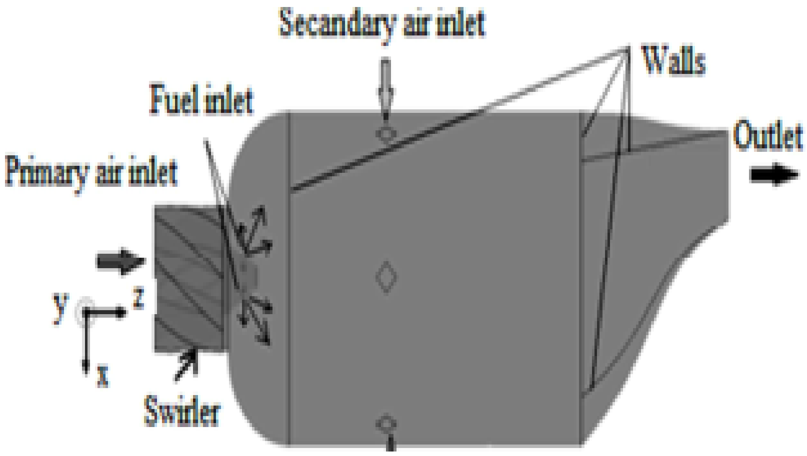

Fig.1 Basic combustion chamber

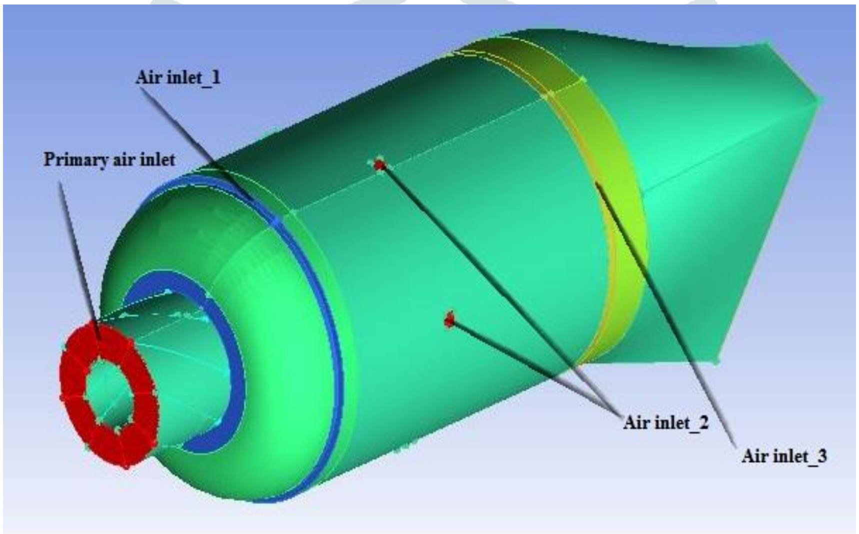

Fig.2 Modified combustion chamber

III. MESHING AND BOUNDARY CONDITIONS

Meshing

The partial differential equations which give solutions for fluid flow and heat transfer is not simple. To achieve solution domains are spitted into subdomains. The governing equations are then discretized and solved inside each subdomain. For this thesis work mesh adaption has done with tetrahedral and prism elements at boundary layer. Meshing for this geometry is done in ICEM CFD, Total elements: 787188, Total nodes: 141051

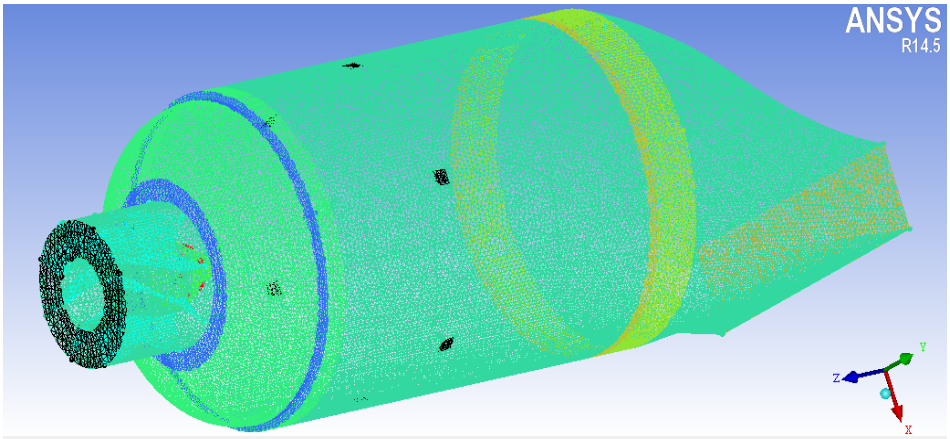

Fig.3 Modified combustion chamber mesh model

Eddy Dissipation Combustion Model

In Eddy Dissipation Combustion model chemical reaction takes place very fast in molecular level relative to transport processes to the flow. When reactants mix at lower molecule level, instantaneously form products. The model assumes reaction rate is directly proportional to time required to mix reactant. In case of turbulent flow reaction rate is directly proportional to Kinetic Energy and Kinetic dissipation. In this model mixing rate is dominated by eddy properties.

Boundary Conditions in ANSYS CFX-Pre

Inlet Air Temperature 300K and 573K, Preheated fuel is used- Modified design

Table.1 Applied boundary conditions

Boundary condition 1	Speed (m/s)
Primary air inlet	10
Fuel inlet	40
Air inlet 1*	10
Air inlet 2	6
Air inlet 3*	6

Boundary condition 2	Speed (m/s)
Primary air inlet	30
Fuel inlet	100
Air inlet 1	30
Air inlet 2	20
Air inlet 3	20

- For preheated fuel inlet temperature will be 573K
- (*) indicates not preferred for basic can type

IV. RESULTS AND DISCUSSIONS

Inlet air stream lines for both basic and modified combustors

Below figures shows how the stream lines are formed along the combustor at same air velocity

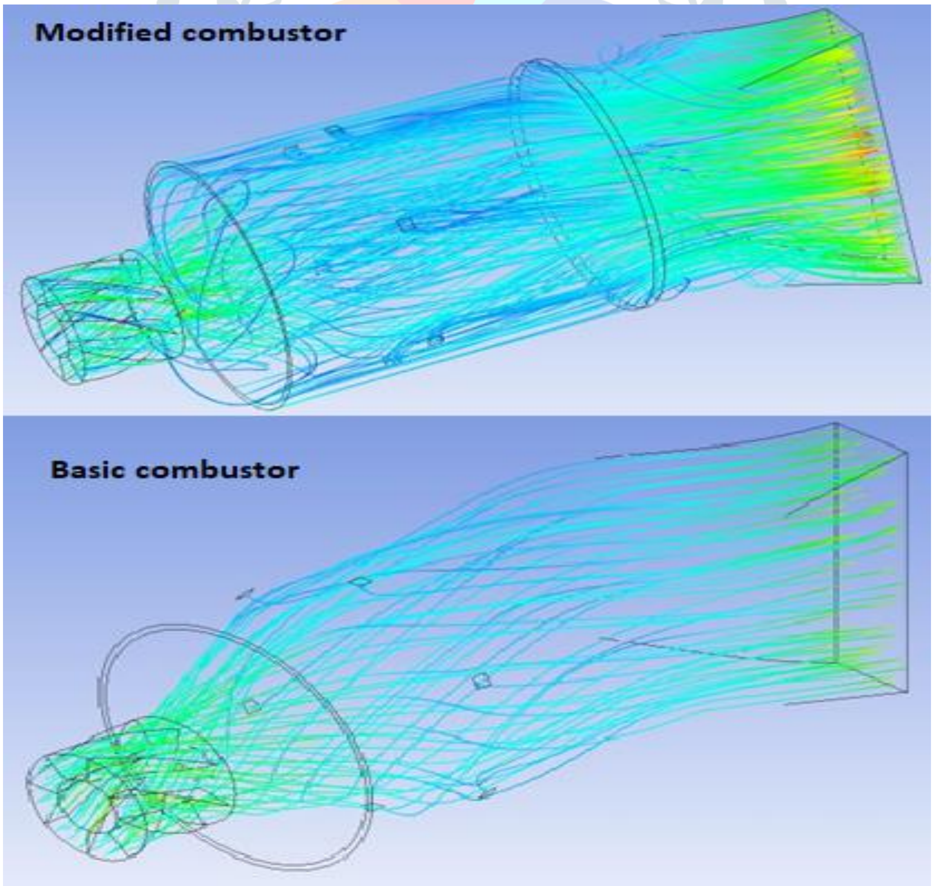

Fig.4 Stream lines path in both Basic and Modified combustor

A. **Test-1** Boundary condition 1 at Air inlet temperature 300K and fuel inlet temperature 300K. In this test we compared the performance of both modified and basic combustors with same boundary conditions

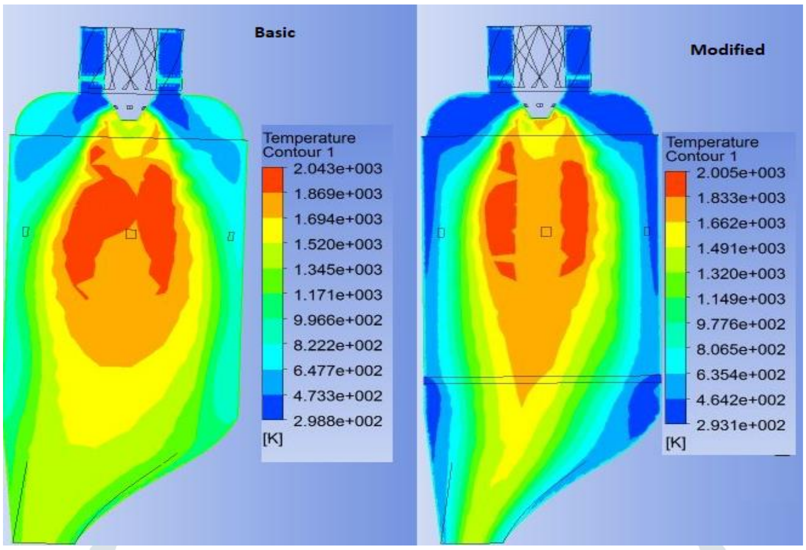

Fig.5 Temperature distribution over basic and modified combustor

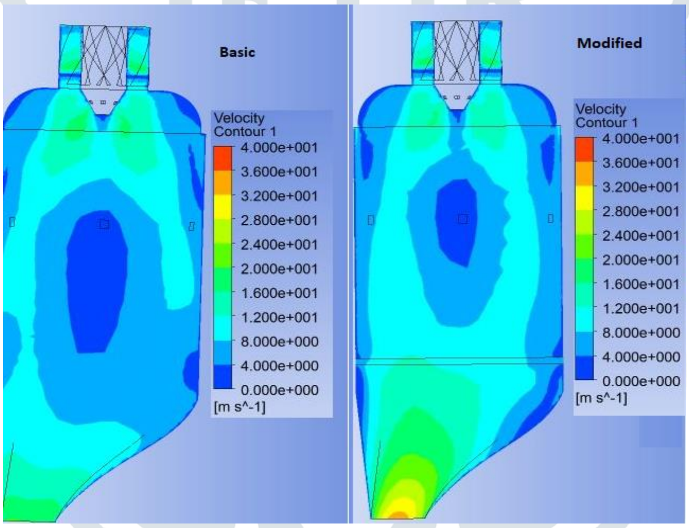

Fig.6 Velocity distribution over basic and modified combustor

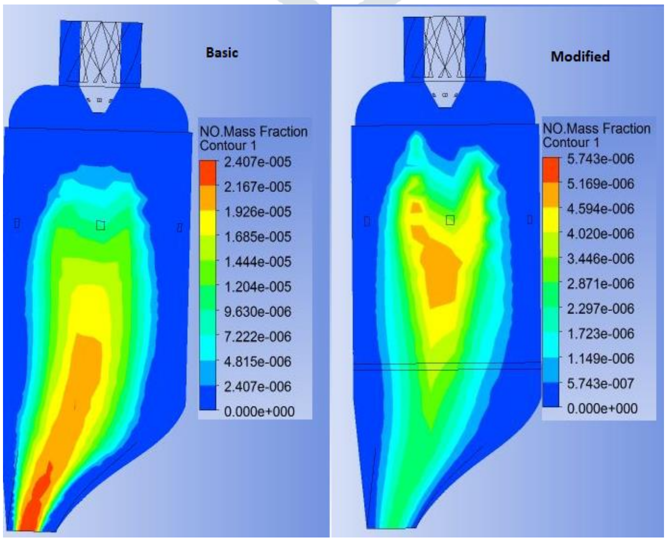

Fig.7 NO mass fraction over basic and modified combustor

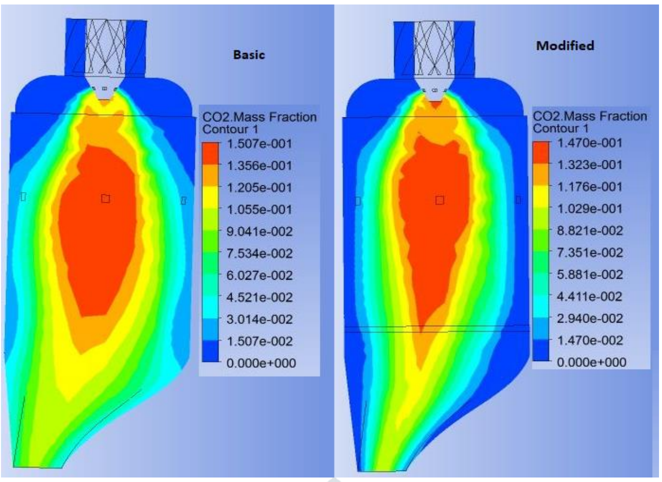

Fig.8 CO_2 mass fraction over basic and modified combustor

From the results in temperature contour we observe, the maximum temperature and average nozzle exit temperature is reduced slightly, wall temperatures are decreased due to slit openings, well cooling along the combustor walls are provided which increases the structural durability of combustor. In velocity contour we observe the velocity is radially increased over the nozzle exit which provides best transfer of heat to turbine blades. Coming to emissions we have CO_2 mass fraction decrement in both maximum and average value and NOx mass fraction is ultimately decreased over the can area in modified combustor.

B. **Test-2** Boundary condition-2 at air inlet temperature 573K and fuel inlet temperature 300K and 573K (preheated). In this test we given the same inlet air temperature as 573K, here we used only modified combustor at different fuel inlet conditions. The main purpose of this simulation is to prove the emissions are also reduced by using preheated fuel injection.

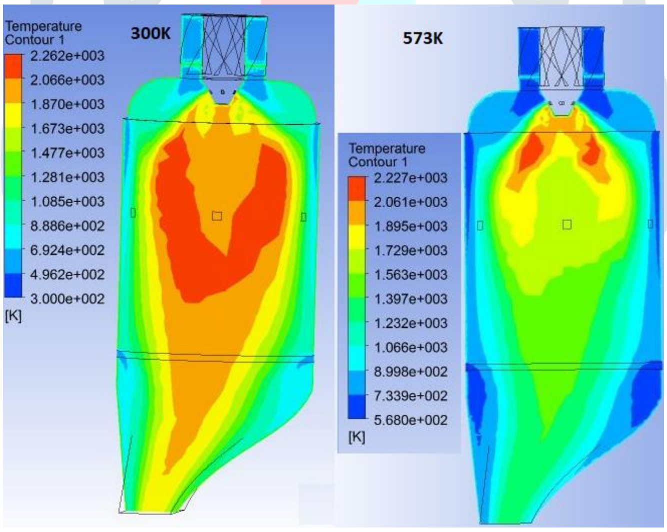

Fig.9 Temperature distribution over normal and preheated condition

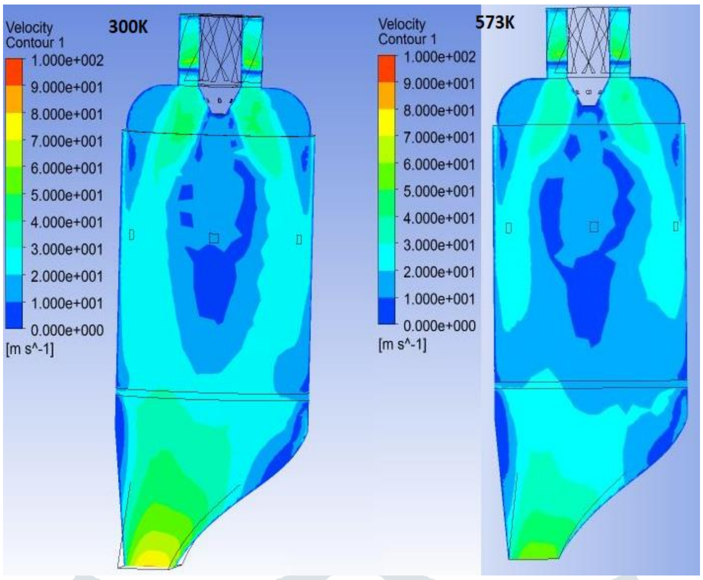

Fig.10 Velocity distribution over normal and preheated condition

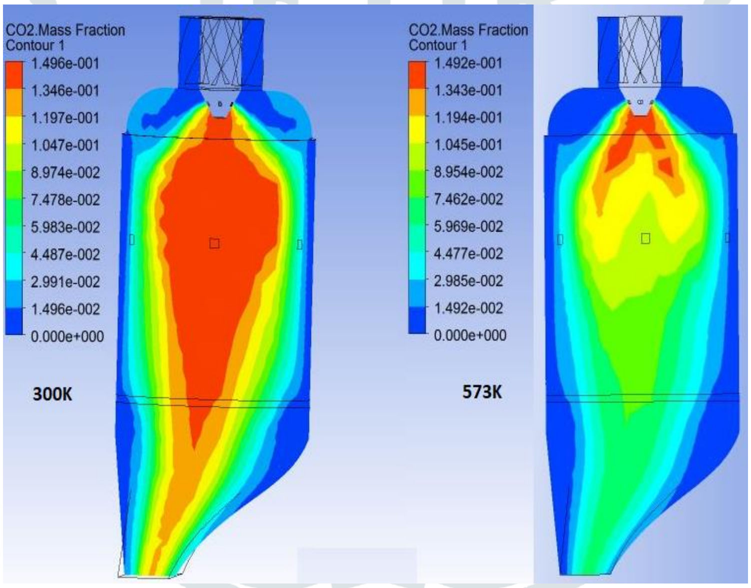

Fig.11 CO_2 mass fraction over normal and preheated condition

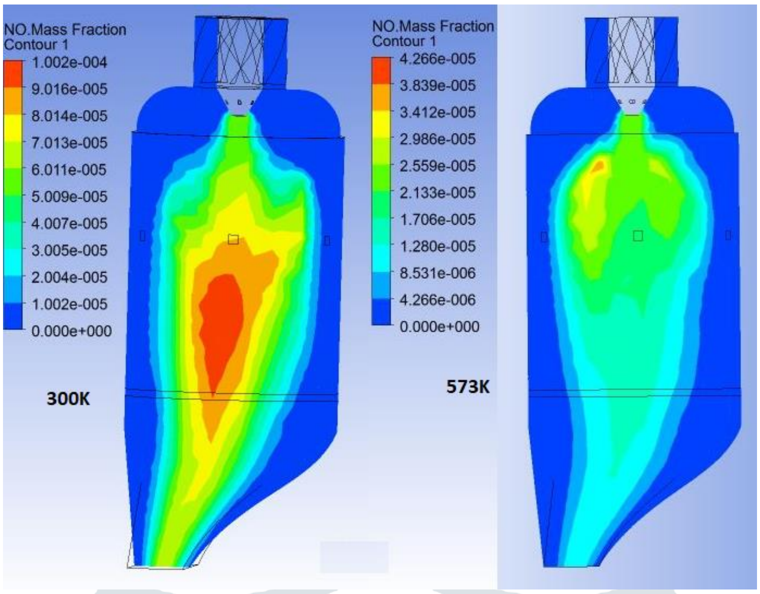

Fig.12 NO mass fraction over normal and preheated condition

Temperature contour- we observe the slight decrement of temperature over the can area in preheated fuel injection when compared to normal fuel injection. Temperature distribution over the can area is perfectly maintained under preheated fuel injection.

Velocity contour- velocity over the nozzle exit area is slightly decreased under preheated fuel injection when compared to normal one i.e., at 300K

Emissions- CO_2 mass fraction is majorly decreased over the can area in preheated fuel injection when compared to normal. NOx mass fraction is highly reduced both in maximum value and over the can area in preheated fuel injection can.

V. CONCLUSION

Methane as fuel the flame temperature produced by the flame with fast combustion has been reached up to 2043K in basic combustor and 2005K in modified at normal atmospheric conditions

From Test-1 boundary conditions are equally applied

Velocity at nozzle exit is radially increased in modified combustor and wall temperature is well cooled which increases the durability and life time of the combustor when compared to basic one. Emissions are mostly reduced in modified combustor which makes it ecologically best.

From Test-2 normal atmospheric and preheated conditions are applied to fuel

Temperature and velocity across the can area in modified combustor is slightly decreased, but in other hand we achieved great reduction in emissions (CO_2 & NOx) which proves preheated fuel injection is ecologically best when compared to normal method.

REFERENCES

[1] Mattingly,J.D, Elements of Gas Turbine Propulsion, McGraw-Hill, NewYork, 1996.pp 65-76, 244-252, 370.
[2] Ganesan, V, Gas Turbines 3E, Tata McGraw-Hill Education, 01 Apr 2010.
[3] Anderson John D, Computational Fluid Dynamics Jr., McGraw – Hill Education (India) Private Limited.pp 38-46, 75-80.
[4] Firoj H Pathan, Nikul K Patel, Mihir V Tadvi,, Numerical Investigation of the Combustion of Methane Air Mixture in Gas Turbine Can Type Combustion Chamber, International Journal of Scientific & Engineering Research, Volume 3, Issue 10, October-2012 (ISSN 2229-5518), Available from URL (https://www.ijser.org/paper/Numerical-Investigation-of-the-Combustion-of-Methane-Air-Mixture-in-Gas-Turbine-Can-Type-Combustion-Chamber.html)