Design and CFD Analysis of Combustion Chamber of Jet Engine to Reduce Formation of NOx

Siddhanth M. Pasalkar[1], **Muzzammil S. Shaikh**[2], **Murtuza M. Pipulyawala**[3]

[1-3] *Student Department of Mechanical Engineering, Rizvi College of Engineering, Mumbai, Maharashtra, India.*

Prof. Husain Jasdanwalla[4]

[4]*Asst. Professor, Department of Mechanical Engineering, Rizvi College of Engineering, Mumbai, Maharashtra, India.*

***

Abstract – *This project emphasizes on the reduction of NO_x in the combustion chamber of the jet engine. Combustion chamber is the main part of the jet engine where many important chemical reactions takes place and generates heat. This paper focuses on the percentage of NO_x in the output and how we can reduce that by changing some parameters into input.*

The challenges in designing high performance combustion systems have not changed significantly over the years, but the approach has shifted towards a more sophisticated analysis process. We first took the standard design i.e. the first design and applied standard boundary conditions, then observed the results carefully. Looking at that we decided to make some changes in the standard design i.e. the second design where we found drastic change in NO_x emission and some more positive changes. This served the need of the purpose. This paper presents the design for combustion chamber and it also include the solid model carried out on SOLIDWORKS and CFD simulation is carried out with the CFD tool ANSYS.

Key Words: **Combustion chamber, Jet engine, Computational fluid dynamics, Ansys Fluent, NO_x reduction.**

1. INTRODUCTION

The combustion chamber of a gas turbine is where the energy that drives the whole system is added. The combustion chamber consists of a cylinder, fuel air mixture passes into the mouth of the cylinder and additional air may pass around the outside of it to keep the cylinder cool. This air is then introduced through holes and slots along the cylinder according to our new design. The main aim of our new design is to reduce the NO_x production.

In gas turbine combustors, the fuel is injected into the combustion chamber through a set of nozzles. The shape and direction of the nozzles and baffles in the combustor are carefully designed to ensure both even mixing and a stable flame within the combustor. The fuel air mixture ignites in the combustion zone, releasing energy as heat.

The air flow through all parts of the combustion chamber must be carefully managed to avoid flame instability and turbulence which will lead to energy loss. The aim is to produce a smooth flow of air.

The addition of air into the combustion chamber is also carefully managed in order to control the production of NO_x during the combustion process. The high temperatures within the combustion zone will lead to ready production of nitrogen oxides from the reaction between oxygen and nitrogen from air. However, our concept of combustors rely on careful mixing of the fuel and air in stoichiometric proportions before the mixture exits the combustor to keep NO_x production under control.

2. LITERATURE REVIEW

2.1. SURVEY: Engine technology has continuously evolved over the last 70 years, and reduction in fuel burn has always been a driving force behind this progress. More fuel-efficient engine cycles, often made possible through the use of new materials, has led to increasing pressures and temperature within the combustor. Since this tends to increase the emissions of nitrogen oxides (NO_x), the control of these emissions through the combustor design is a significant challenge. The ICAO regulatory limits for engine NO_x emissions has been gradually tightened over time, and are usually referred to by the corresponding CAEP meeting number (CAEP/2, CAEP/4, CAEP/6 and CAEP/8). The engine NO_x standard, and the new aeroplane CO_2 standard, contribute in defining the design space for new products so as to address both air quality and climate change issues

2.2. NEEDS TO CONTROL NO_x: NO_x represents a family of seven compounds. NO_x in the atmosphere that is generated by anthropogenic (human) activities. NO_2 is not only an important air pollutant by itself, but also reacts in the atmosphere to form ozone (O_3) and acid rain. It is important to note that the ozone that we want to minimize is tropospheric ozone; that is, ozone in the ambient air that we breathe. We are not talking about stratospheric ozone in the upper atmosphere that we cannot breathe. Stratospheric ozone protects us and the troposphere from ionizing radiation coming from the sun. Tropospheric ozone has been and continues to be a significant air pollution problem and is the primary constituent of smog. NO_2 reacts in the presence of air and ultraviolet light (UV) in sunlight to form ozone and nitric oxide (NO). The NO then reacts with free radicals in the atmosphere, which are also created by the UV acting on volatile organic compounds (VOC). The free radicals then recycle NO to NO_2. In this way, each molecule of NO can produce ozone multiple times. This will continue until the VOC are reduced to short chains of carbon compounds that cease to be photo reactive (a reaction caused by light). A VOC molecule can usually do this about 5 times. In addition to the NO_2 and Ozone concerns, NO_x and sulphur oxides (SO_x) in the 2 atmospheres are captured by moisture to form acid rain. Acid rain, along with cloud and dry deposition, severely affects certain ecosystems and directly affects some segments of our economy. All of these facts indicate an obvious need to reduce NO_x emissions. However, to successfully do so, we must understand the generation and control of the NO_x family of air pollutants.

3. DESIGN DETAILS OF THE MODEL:

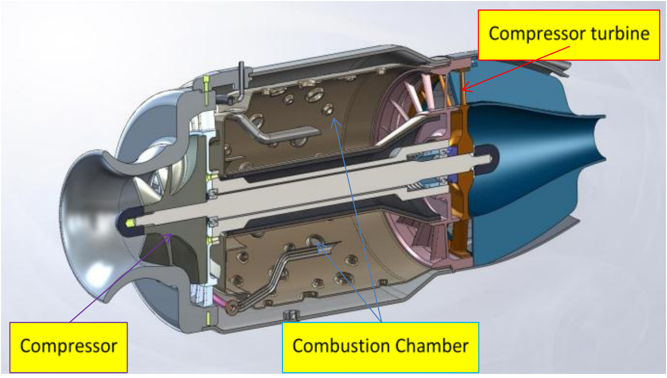


Fig. 1- 3D MODEL OF STANDARD DESIGN

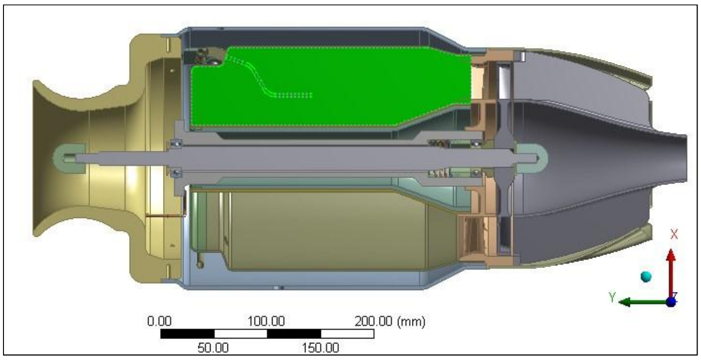


Fig. 2- 2D MODEL OF STANDARD DESIGN

We have used CAD and Ansys Fluent for designing and analysis of the model in our project.

We have considered a 2-dimension geometry. The first design which we have selected is the standard one. The cross-sectional view of the standard turbine consists of compressor turbine, compressor and combustion chamber as shown in the figure.

In the first design as we see the fuel inlet pipe is at the middle and the air inlet is given at the beginning of the combustion chamber i.e. the left wall. Fine meshing is done wherever needed i.e. near the inlet pipe and the air inlet. Hexa modelling and map mesh is used in meshing. As this geometry has infinite particles so if we have infinity in the equation, we can solve it. In the Governing, Momentum, or Energy equation. If we have infinity, we cannot solve it, we need to convert the infinite domain into finite domain, we need to discretize into elements, this is why meshing is done. Division control are given at the entry of inlet, 10 divisions are given and fine mesh is provided for better results.

We have considered viscous model as k-epsilon model with standard wall function model which is the standard one. Material properties or material the temperature distribution is also not appropriate as seen.

Considering all these flaws, we made a few changes in the design i.e. Design-2. The size of the air inlet is reduced from the left wall and some porous air inlet are introduced on the top wall. The fuel inlet is opened from the top left wall considering the flow of air and fuel will properly be mixed.

In the second design as we see the fuel inlet is at the top left wall and the air inlet positions are changed. Fine meshing is done wherever needed i.e. the inlet pipe and the air inlet. Hexa model and map meshing is used. All boundary conditions are kept same as Design-1 and analysis is seen again for 200 iterations.

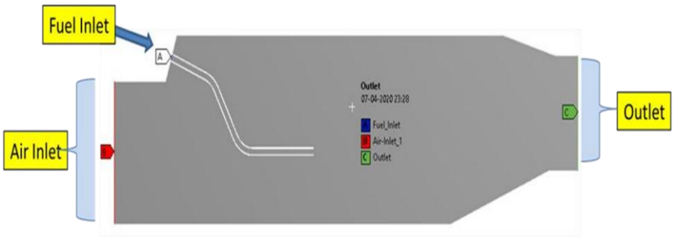


Fig. 3- DESIGN 1 (STANDARD DESIGN)

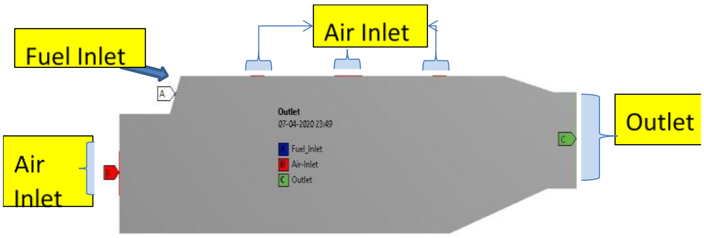


Fig. 4- DESIGN 2 (IMPROVED DESIGN)

selected is Methane-Air mixture. Method used for solving is coupled system method i.e. Pressure-Velocity. Momentum higher order, pressure higher order 2nd, pseudo transient 2nd order as shown in the screenshot. As first we had considered 150 iterations then too there were some fluctuations, therefore we increased it too 200 iterations and saw stabilized lines till 180-190 iterations. After getting the results, we observed that N_2 developed near the mixture is not properly

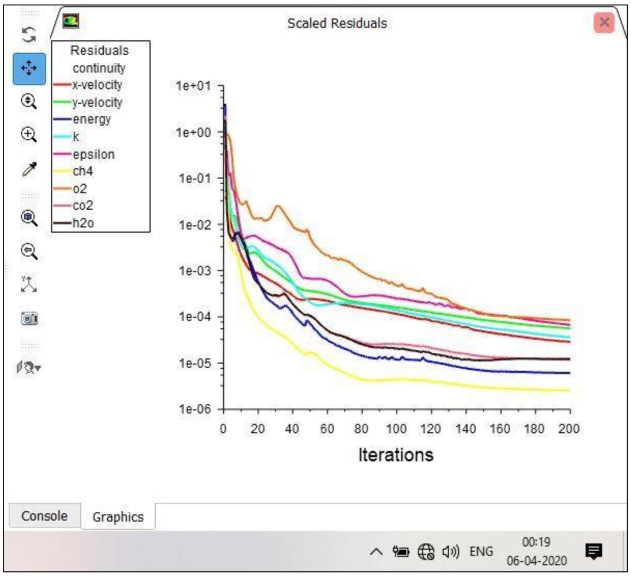


Fig. 5- ITERATION GRAPH

After 180 iteration the results observed to be stabilize and hence 200 iterations are good enough for this analysis.

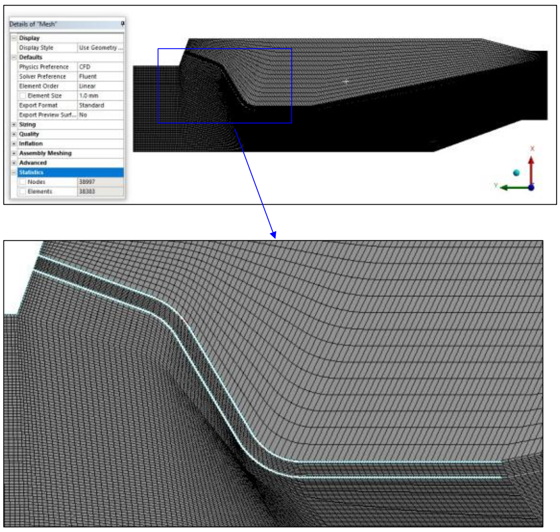
4. OBJECTIVE:

- The emission of the NO_x should be reduced.
- The efficiency of engine should increase.
- The distribution of heat should be proper and there should be heat concentration zones.
- The air fuel mixture should be adequate before combustion.
- To maximise the enthalpy.

5. BOUNDARY CONDITIONS:

Below are some detailed photos of Boundary conditions applied during the analysis.

Meshing :

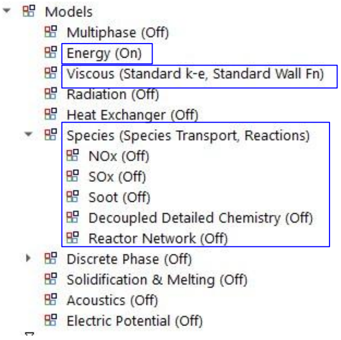


Details of "Mesh"

Display	
Display Style	Use Geometry ...
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	1.0 mm
Export Format	Standard
Export Preview Surf...	No
Sizing	
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	
Nodes	38997
Elements	38383

Hex element with map mesh used and also make sure the very fine mesh at air/fuel entry for better accuracy.

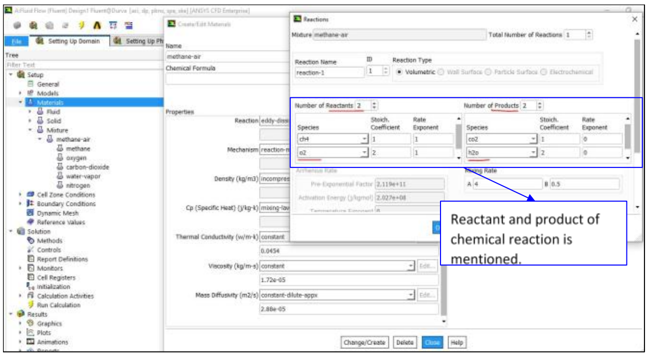
- **Fluid Model Considered:**



- Models
  - Multiphase (Off)
  - Energy (On)
  - Viscous (Standard k-e, Standard Wall Fn)
  - Radiation (Off)
  - Heat Exchanger (Off)
  - Species (Species Transport, Reactions)
    - NOx (Off)
    - SOx (Off)
    - Soot (Off)
    - Decoupled Detailed Chemistry (Off)
    - Reactor Network (Off)
  - Discrete Phase (Off)
  - Solidification & Melting (Off)
  - Acoustics (Off)
  - Electric Potential (Off)

Highlighted equations in blue box are considered.

Material Properties:



Reactions

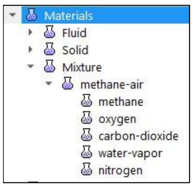
Mixture: methane-air
Total Number of Reactions: 1
Reaction Name: reaction-1
ID: 1
Reaction Type: ● Volumetric ○ Wall Surface ○ Particle Surface ○ Electrochemical
Number of Reactants: 2
Number of Products: 2

Species	Stoich. Coefficient	Rate Exponent
ch4	1	1
o2	2	1

Species	Stoich. Coefficient	Rate Exponent
co2	1	0
h2o	2	0

Arrhenius Rate
Pre-Exponential Factor: 2.119e+11
Activation Energy (j/kgmol): 2.027e+08
Mixing Rate
A: 4
B: 0.5

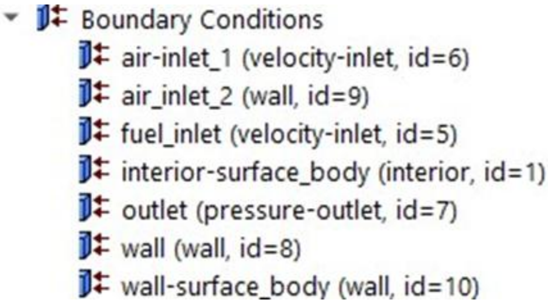
Reactant and product of chemical reaction is mentioned.



- Materials
  - Fluid
  - Solid
  - Mixture
    - methane-air
      - methane
      - oxygen
      - carbon-dioxide
      - water-vapor
      - nitrogen

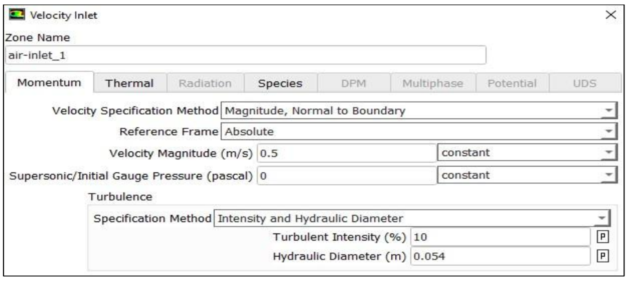
Methane-Air mixture is taken in which 4 gases (CH_4, O_2, CO_2, H_2O) are taken in consideration.

Boundary conditions :

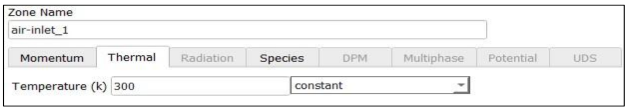


- Boundary Conditions
  - air-inlet_1 (velocity-inlet, id=6)
  - air_inlet_2 (wall, id=9)
  - fuel_inlet (velocity-inlet, id=5)
  - interior-surface_body (interior, id=1)
  - outlet (pressure-outlet, id=7)
  - wall (wall, id=8)
  - wall-surface_body (wall, id=10)

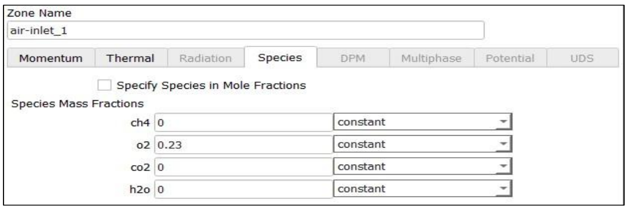
Air inlet: Velocity inlet- 0.5 m/s & 300K temp:



Zone Name: air-inlet_1
Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude (m/s): 0.5 constant
Supersonic/Initial Gauge Pressure (pascal): 0 constant
Turbulence
Specification Method: Intensity and Hydraulic Diameter
Turbulent Intensity (%): 10
Hydraulic Diameter (m): 0.054

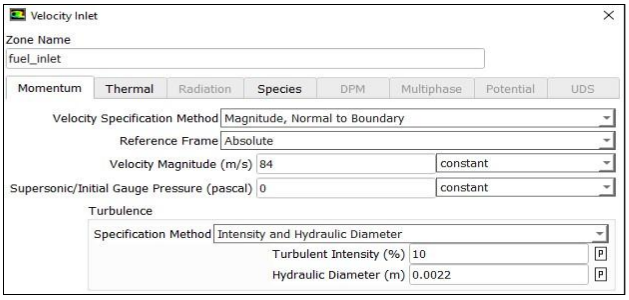


Zone Name: air-inlet_1
Temperature (k): 300 constant



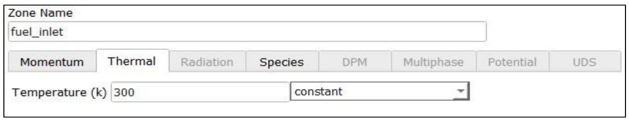
Zone Name: air-inlet_1
☐ Specify Species in Mole Fractions
Species Mass Fractions
ch4: 0 constant
o2: 0.23 constant
co2: 0 constant
h2o: 0 constant

Fuel inlet: Velocity inlet- 84 m/s & 300 K temp. :



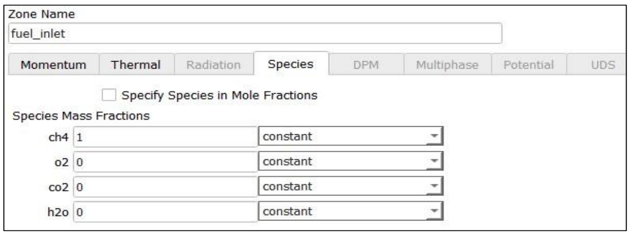
Zone Name: fuel_inlet

Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude (m/s): 84 constant
Supersonic/Initial Gauge Pressure (pascal): 0 constant
Turbulence
Specification Method: Intensity and Hydraulic Diameter
Turbulent Intensity (%): 10
Hydraulic Diameter (m): 0.0022



Zone Name: fuel_inlet

Temperature (k): 300 constant

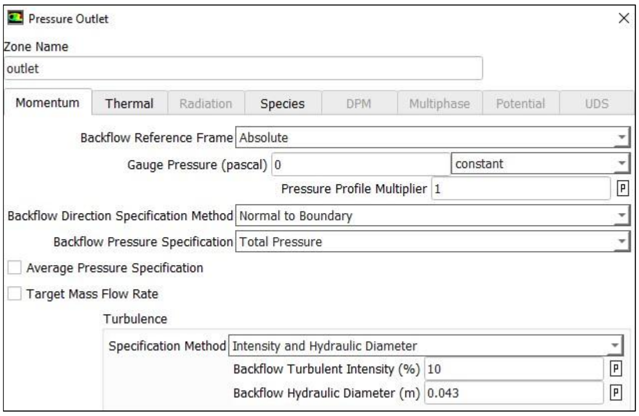


Zone Name: fuel_inlet

☐ Specify Species in Mole Fractions

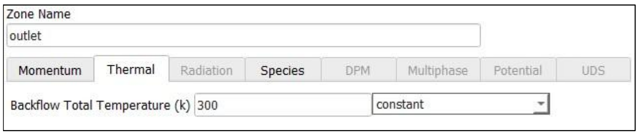
Species Mass Fractions
ch4: 1 constant
o2: 0 constant
co2: 0 constant
h2o: 0 constant

Outlet: Defined as pressure outlet: 0(zero) gauge pressure with 300 K temp. :



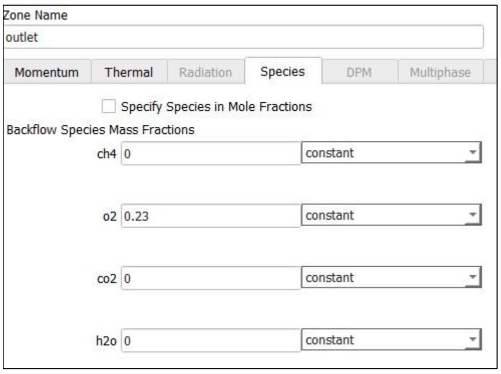
Zone Name: outlet

Backflow Reference Frame: Absolute
Gauge Pressure (pascal): 0 constant
Pressure Profile Multiplier: 1
Backflow Direction Specification Method: Normal to Boundary
Backflow Pressure Specification: Total Pressure
☐ Average Pressure Specification
☐ Target Mass Flow Rate
Turbulence
Specification Method: Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%): 10
Backflow Hydraulic Diameter (m): 0.043



Zone Name: outlet

Backflow Total Temperature (k): 300 constant

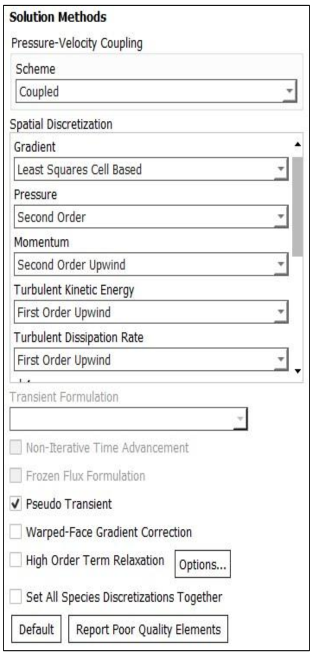


Zone Name: outlet

☐ Specify Species in Mole Fractions

Backflow Species Mass Fractions
ch4: 0 constant
o2: 0.23 constant
co2: 0 constant
h2o: 0 constant

Solver setting : Pressure Velocity coupling considered :



Solution Methods

Pressure-Velocity Coupling

Scheme: Coupled

Spatial Discretization

Gradient: Least Squares Cell Based

Pressure: Second Order

Momentum: Second Order Upwind

Turbulent Kinetic Energy: First Order Upwind

Turbulent Dissipation Rate: First Order Upwind

Transient Formulation

☐ Non-Iterative Time Advancement

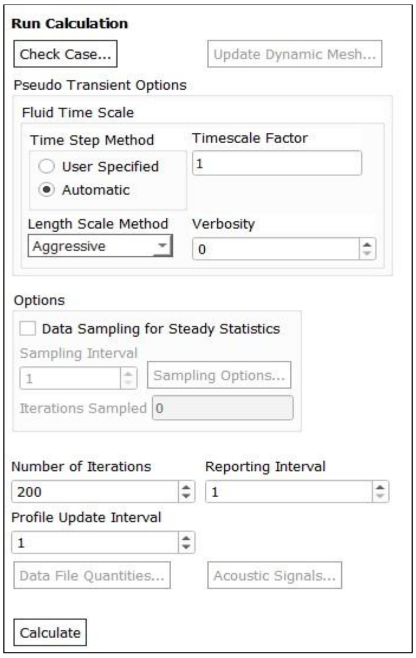
☐ Frozen Flux Formulation

☑ Pseudo Transient

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation

☐ Set All Species Discretizations Together



Run Calculation

Pseudo Transient Options

Fluid Time Scale

Time Step Method: ○ User Specified ● Automatic

Timescale Factor: 1

Length Scale Method: Aggressive

Verbosity: 0

Options

☐ Data Sampling for Steady Statistics

Sampling Interval: 1

Iterations Sampled: 0

Number of Iterations: 200

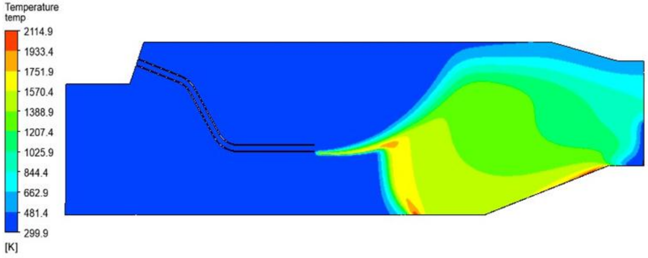
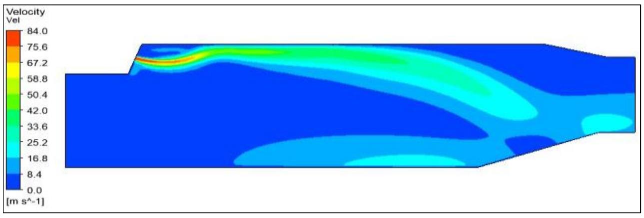
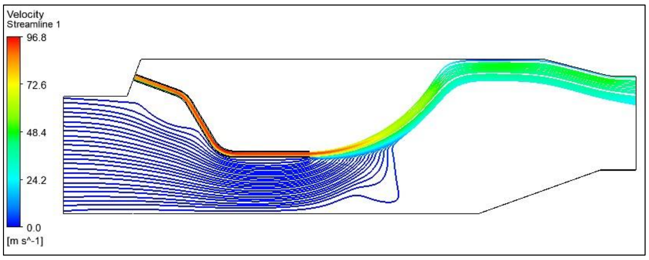
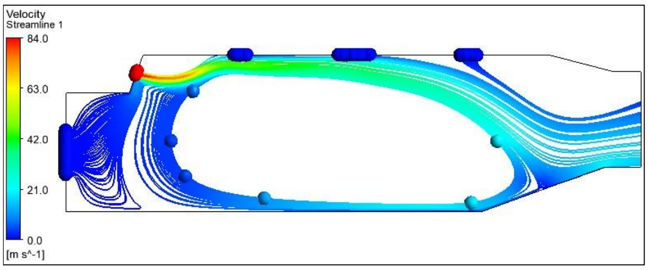
Reporting Interval: 1

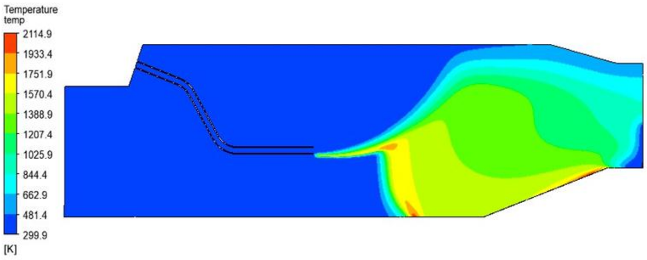
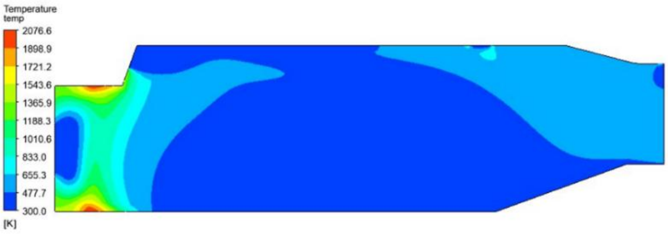
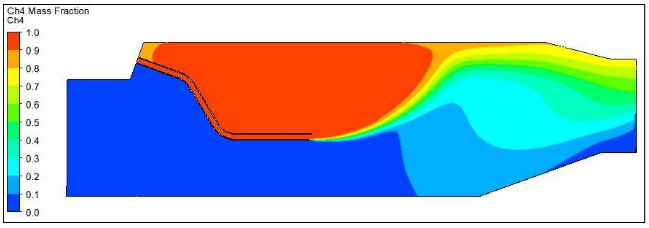
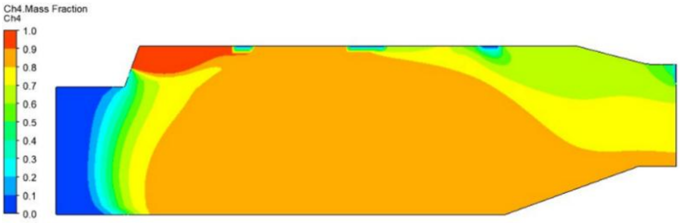
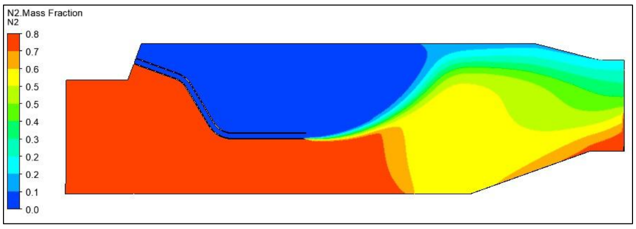
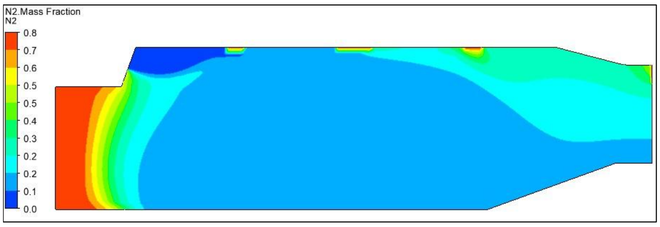
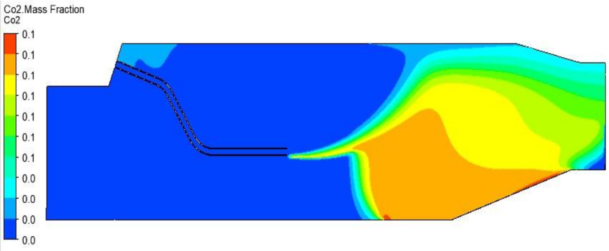
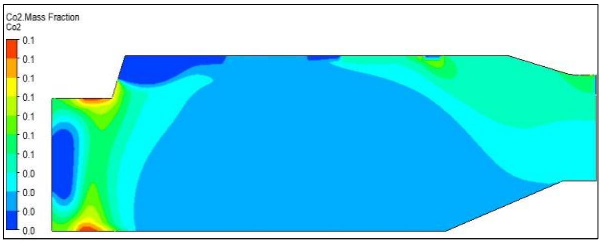
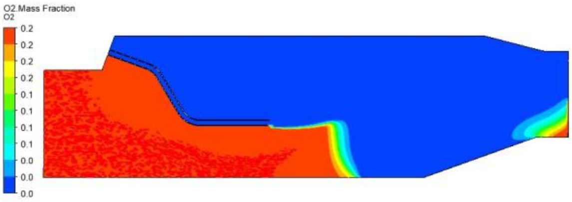
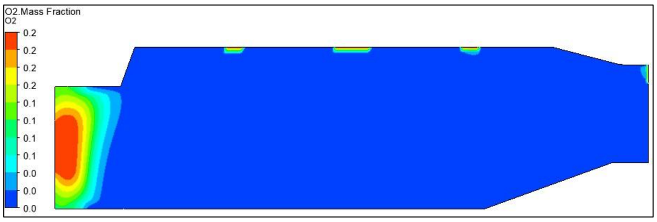
Profile Update Interval: 1

200 iterations are taken in for the graph to be stable. Same boundary conditions are applied in the Design 2.

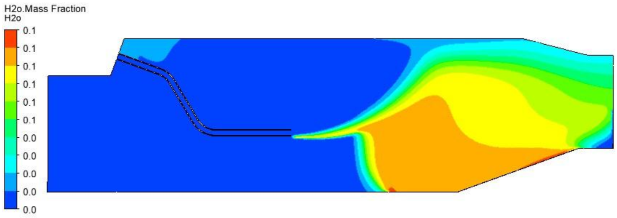
6. ANALYSIS COMPARSION OF DESIGN 1 & DESIGN 2 (IN DIFFERENT PARAMETERS) :

(RESULT COMPARISON OF DESIGN 1 & 2)

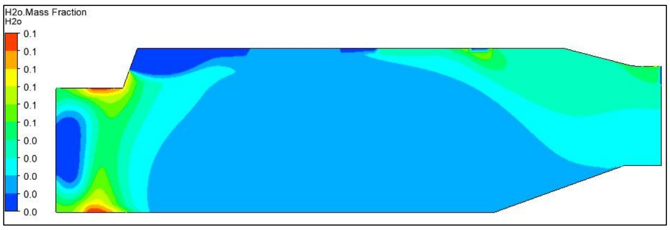
`DESIGN 1	DESIGN -2
Velocity Contour Plot :  	○ **Velocity Contour Plot:** 
○ **Velocity Streamline (path) Plot:**  	○ **Velocity Streamline (path) Plot:** 

○ **Temperature plot:**  	○ **Temperature plot :** 
○ **CH4 Mass Fraction plot:**  	○ **CH_4 Mass Fraction Plot :** 
○ **N2 Mass Fraction plot:**  	○ **N2 Mass Fraction plot:** 
○ **Co2 Mass Fraction plot:**  	○ **CO_2 Mass fraction plot :** 
O2 Mass Fraction plot:  	○ **O2 Mass Fraction plot:** 

H2o Mass Fraction plot:



○ **H2o Mass Fraction plot:**



7. DISCUSSION OF THE RESULTS:

It has been found that change in the dilution holes area and reduction in air entry for Design 2 gives 38% reduction in total temperature before the turbine as well as approx. 55% reduction in emissions of NO_x and 40% reduction of CO emissions. It is therefore possible to develop several variants of controlling the changes of the dilution holes area of the combustion chamber for the desired effect (maximizing the enthalpy, reducing emissions of harmful compounds). Three-dimensional maps of the total temperature for the combustion chamber outlet and the emission of NO_x and CO can be used to develop the right control of variable area of the dilution holes zone. Control of the geometry of the combustion chamber is also very attractive because of the increase in operability of the engine. By controlling the geometry of the combustion chamber one can easily control the amount of air in the primary zone, thereby regulating the stoichiometric factor. As a result, it is possible to prevent the flame blow off in the transient engine operating conditions, i.e. with a sudden reduced / increased amount of fuel. In the case of a sudden reduction in the amount of fuel, a gas turbine rotor having a large moment of inertia continues to provide a lot of volume of air to the combustion chamber, which in combination with a low dose of fuel can in some cases lead to exceeding the lower flammability limit and flame blow off. In these situations, the rapid opening of the holes of the secondary zone would help in providing a larger amount of air to the dilution zone of the combustion chamber passing at the same time to the primary zone. A similar situation could occur for a sudden increase of fuel delivery.

8. CONCLUSIONS:

From CFD results show that proposed concept of geometry combustor gives 38% reduction in total temperature at combustor outlet. Additionally, 55% NOx and 40% CO emission reductions were obtained. Active control of the combustion chamber geometry can lead to increase in engine operability. Variable dilution holes area can act as an active anti-surge system as well as lean blowout control system. The three-dimensional maps of the total temperature and the emissions of NO_x and CO, obtained in the experiment at combustor outlet, can be used to develop the control system of variable area of dilution holes.

REFERENCES:

1. Raminder Singh, B. Dinesh Kumar, Sai Kumar, “Design and CFD Analysis of Gas Turbine Engine Chamber”, IJSR, 2013, India.
2. K. V. Chaudhari, D. B. Kulshreshtha, S.A. Channiwala, “Design and CFD Simulation of Annular Combustion Chamber with Kerosene as Fuel for 20 kW Gas Turbine Engine”, IJERA, 2012, India.
3. Lucilene O. Rodrigues, Harley S. Alencar, Marco A. R. Nascimento, Osvaldo J. Venturini “AERODYNAMIC ANALYSIS USING CFD FOR GAS TURBINE COMBUSTION CHAMBER”, Research Gate,2007,Brazil.

4. Prithwish Kundu, “Gas Turbine Combustion Chamber Design for Viscous Fuels”, 2012, North Carolina.
5. C. Priyant Mark, A. Selwyn, “Design and analysis of annular combustion chamber of a low bypass turbofan engine in a jet trainer aircraft”, science direct, 2016, India.
6. The Jet Engine, Fifth edition, Rolls Royce.
7. Aircraft Gas Turbine Power plants, Charles E.
8. Otis & Peter A. Vosbury.
9. The Aircraft Gas Turbine Engine and its Operation, Pratt & Whittney.