iJRASET
International Journal For Research in
Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

**Volume:** CAAA-2018 **Issue:** onference **Month of publication:** April 2018

**DOI:**

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

# Comparative Study on Trapped Vortex Combustor with Normal Combuster

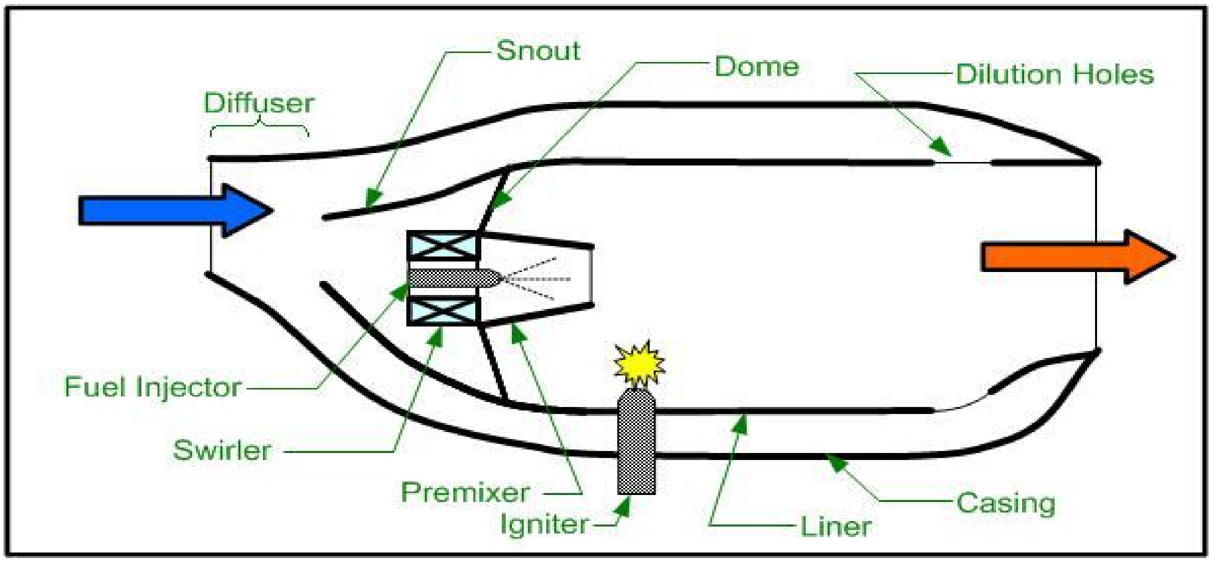
Mr. S. Balaji[1], Asst Prof Aero[2], N. Rajadurai[3], J. Veema[4], J. Poornima[5], P. Deepa[6]
[1, 2, 3, 4, 5, 6] *Department of Aeronautical Engineering, Nehru institute of engineering and technology,Coimbatore-105*

***Abstract: In gas turbine engines, the combustion system is the main component where the efficiency of the engine is evaluated. High combustion efficiency, stability of the combustion flame and low emissions of toxic gases are the main problems raised in the combustion chamber. Recently, combustion instability is given more importance as it is one of the problems which is affecting the performance of the gas turbine engine. More research work has been focused on the combustion stability and the ways to improve it. In engines, since most flowing fuel-air mixtures are turbulent and turbulence is given prime importance as it is known to enrich flame speed and fuel-air mixture considerably. Flame speed can be appreciably increased by turbulence, as evidenced by the very high burning rates achieved in gas turbine engines. Design and performance of a combustor is strongly affected by the aerodynamic processes. The achievement of aerodynamically efficient designs characterized by good mixing and stable flow patterns with minimal parasitic losses is one of the primary design objectives.***

***Trapped Vortex Combustor (TVC) design is based on mixing hot combustion products and reactants at a high rate by a cavity stabilization concept. By creating a vortex/eddies which is more or less like turbulent mixing inside the combustion chamber, proper mixing of fuel-air can be obtained. Turbulence occurring in a TVC combustion chamber is "trapped" within a cavity where reactants are injected and efficiently mixed. Under reacting flow conditions, the flow in the cavity region has the flow structure which is highly dependent on the momentum flux ratio between the cavity and the mainstream flow. Since, part of the combustion occurs within the recirculation zone, a "typically" flameless regime can be achieved, while a trapped turbulent vortex may provide significant pressure drop reduction. Besides this, TVC is having the capability of operating as a staged combustor if the fuel is injected into both the cavities and the main airflow. Generally, staged combustion systems are having the potential of achieving about 10% to 40% reduction in NOX emissions. In general, if the mixture of fuel-air is proper at the initial stage of the combustion chamber, the level of emissions can be reduced, the combustion stability is obtained through good flow patterns in TVC because of turbulent mixing and this will lead to high combustion efficiency automatically.***

## I. INTRODUCTION

A combustor is a component or area of a gas turbine, ramjet, or scramjet engine where combustion takes place. It is also known as a burner, combustion chamber or flame holder. The combustion chamber can be divided into two areas: the primary zone and the secondary zone. The primary zone is where the majority of the fuel combustion takes place. The fuel must be mixed with the correct amount of air so that a stoichiometric mixture is present. In the secondary zone, unburned air is mixed with the combustion products to cool the mixture before it enters the turbine. In some design, there is an intermediate zone where help secondary zone to eliminate the dissociation products and burn-out soot. 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. 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. Early gas turbine engines used a single chamber known as a can type combustor. Today three main configurations exist: can, annular and annular (also referred to as can-annular tuba-annular). Afterburners are often considered another type of combustor. Combustors play a crucial role in determining many of an engine's operating characteristics, such as fuel efficiency, levels of emissions and transient response . The majority of the combustors are developed base on diffusion flames as they are very stable and fuel flexibility option. In a diffusion flame, there will be always stoichiometric regions regardless of overall stoichiometry. The main disadvantage of diffusion-type combustor is the emission as high temperature of the primary zone produced larger than 70 ppm NOX in burning natural gas and more than 100 ppm for liquid fuel . Several techniques have been tried in order to reduce the amount of NOX produced in conventional combustors. In general, it is difficult to reduce NOX emissions while maintaining a high combustion efficiency as there is a tradeoff between NOX production and CO/UHC production. A combustor concept that is being developed with these goals in mind is the Trapped Vortex Combustor (TVC). This development program will take the TVC from the laboratory test stage through design and fabrication of engine. The Ultra-Compact Combustor (UCC) that operates as an Inter-turbine Burner (ITB) situated in between the high and low pressure turbine stages is modeled with the Trapped Vortex Combustor (TVC) with a single vane containing a notch and with various protuberance designs and arrays, located in the vane and in the TVC. The steady three dimensional governing equations of continuity, momentum, energy, turbulence, and species. fuel-air distribution and temperature distribution within the cavity of an axisymmetric trapped vortex combustor (TVC) under reacting flow condition. k- ε model was used for turbulence modeling while combustion was simulated using the Eddy Dissipation Model. Flame stabilization is achieved by using a cavity, formed by mounting two discs in tandem manner. A self-sustained flame, fuel and air are injected directly into the cavity .The diffusion and premixed flame are two main type of combustion, which are using in gas turbines. Apart from type of flame, there are two kind of combustor design, annular and tubular. The annular type mostly recommended in the propulsion of aircraft when small cross section and low weight are important parameters. Can or tubular combustors are cheaper and several of them can be adjusted for an industrial engine identically. Although there are different types of combustors, but generally, all combustion chambers have a diffuser, a casing, a liner, a fuel injector and a cooling arrangement.



This cavity injections hound be made in such a way it reinforces the trapped vortex in addition to enhancing the mass and energy transport between the cavity and the mainstream flow

As a result, the performance parameters of the TVC such as combustion efficiency, flame stability, etc. will get enhanced. Besides this, TVC is having the capability of operating as a staged combustor. Generally, staged combustion systems are having the potential of achieving about 10% to 40% reduction in NOX emissions. In a TVC the combustion processes is not necessarily completed within the cavity itself, but it may continue downstream when the fraction of unburned gases mixes with fresh mainstream oxidizer. When this happens, this is a staged combustion process, corresponding to a Rich Quench Lean strategy, well known from tests to lower emissions.

## II. OBJECTIVE

The objective of this thesis is to develop a methodology for the Trapped vortex combustor. The purpose is to update the literature and to provide the reader with a simple algorithm that can be used to produce a stable, efficient and feasible design of Trapped vortex combustor. The specific task necessary to accomplish the objectives are

1) Increase residence time, while keeping the pressure drop low.
2) Increase combustion efficiency up to at least 98%.
3) Lower NOX emissions.

Objectives are in place to improve altitude relight, lean-blow-out, and durability while maintaining cost, weight and combustor exit temperature profile. The fall and air to be completely mixed without any loss of fraction. The motive reduce NOX and SOX emission from it.

## III. DESIGNING OF TRAPPED VORTEX COBUSTOR

In the numerical simulation of trapped vortex combustor we taking one traditional combustor and we designed two different trapped vortex combustor by design software CATIA. We design the three types of combustor. They are as follow as

1) Model 1 - Normal combustor.
2) Model 2 - Trapped vortex combustor (inside block).
3) Model 3 - Trapped vortex combustor (outside block).

### *A. Combustor Design*

The simultaneous involvement of evaporation, turbulent mixing, ignition, and chemical reaction in gas turbine combustion is too complex for complete theoretical treatment. Instead, large engine manufacturers undertake expensive engine development programs to modify previously established designs through trial-and-error. They also develop their own proprietary combustor design rules from the experimental results of these programs. These design rules provide a means of specifying the combustor geometry to meet a set of requirements at the given inlet conditions. Numerous published empirical, semi-empirical, and analytical tools have been developed to reduce the need for costly experiments.

Several authors have published preliminary designs for conventional aircraft gas turbines that provide a starting point for the development of modern industrial combustors (Sawyer, 1985;

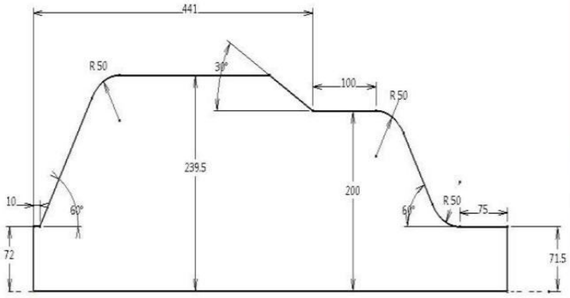
Dodd’s & Bahr, 1990; Mattingly et al. ,2002). The approaches made by these authors differ mainly by the specification of combustor size.

The exact model of the combustor with dimensions and specification was taken from the Marc.R.J.Charest (2006) and redrawn using CATIA and meshed with ANSYS workbench. Flow simulation was performed using ANSYS Fluent.

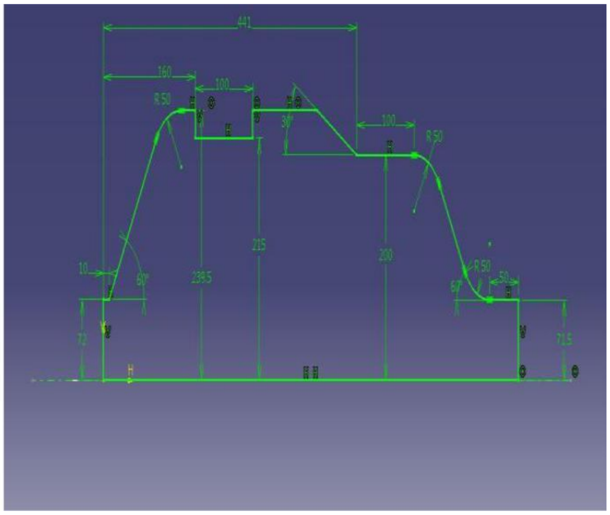
Assumptions made in simplifying the internal flow path of the combustor liner,

1) As the flow inside the whole combustor is complex, only the flow inside the liner alone modeled and uncoupled from the annulus flow.
2) At the combustor swirler downstream the main inlet of the liner is placed. Flow exiting the swirler is assumed as uniform and follows the blade.
3) As the combustor liner model is axisymmetric, a 36 degree periodic model was considered to reduce the mesh count and solving time.
4) The periodic model was chosen carefully so that the dilution holes and swirler vanes should be inside the flow domain.

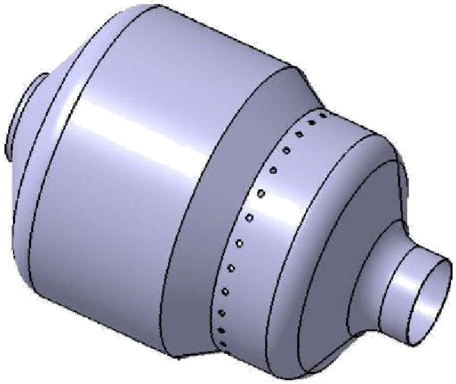
The dimensions of the normal combustor is shown in Figure 4.1. Based on the dimension the initial sketch was created in CATIA and then the solid model was created using shaft command in part design module of CATIA. The combustor liner model is shown in Figure 4.4. Flow path of the combustor liner was modeled using for specified dimension as shown in Figure 4.1. The flow path of the model was simplified as a 36 degree periodic model for flow analysis as shown in Figure. 4.5. Same way the flow path of other combustor model also created.

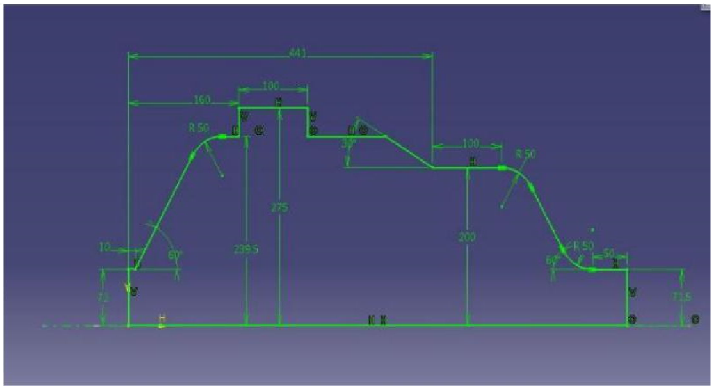


Model 1 2D Normal combustor



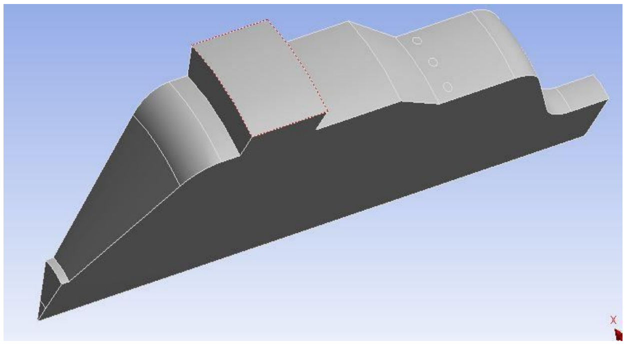
Model 2 2D Trapped vortex combustor(in side)



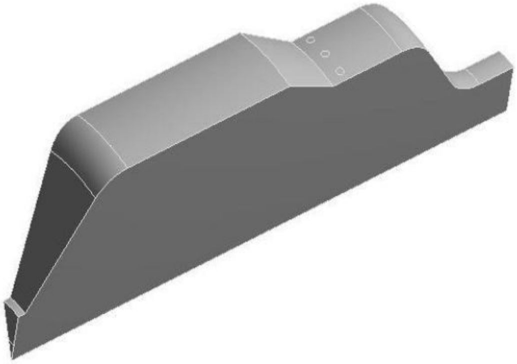


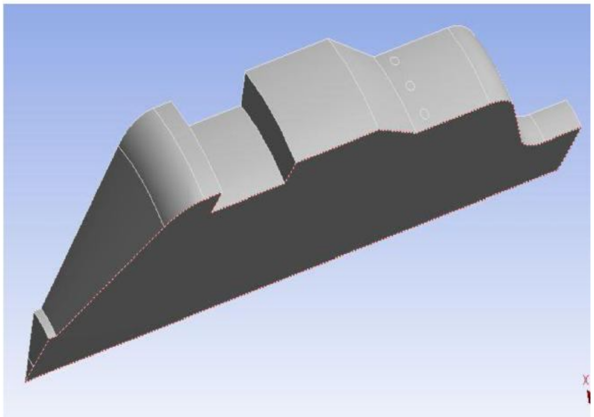
3D Combustion chamber model

Model 3 2D Trapped vortex combustor(outside)



Periodic flow domain of trapped vortex combustor model 2





Periodic flow domain of normal combustor

periodic flow domain of vortex combustor model 1

## IV. NUMERICAL FLOW SIMULATION OF COMBUSTOR MODEL

### A. *Flow Analysis*

Analysis begins with a mathematical model of a physical problem using CFD. It deals with conservation of matter, momentum, and energy that must be satisfied throughout the region of interest. Fluid properties are modeled empirically and simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two-dimensional). Also it provides appropriate initial and boundary conditions for the problem.

### *B. Introduction to cfd:*

CFD is a science of predicting fluid flow, heat and mass transfer, chemical reaction and related performance by solving the set of governing mathematical equations of fluid dynamics the continuity, momentum and energy equations. These equations speak physics.

### *C. Numerical Discretization Techniques*

In this process of numerical discretization each term within a partial differential equation is translated into a numerical analogue that the computer can be programmed to calculate. Some of the numerical discretization techniques.

*1)* Finite Difference method
*2)* Finite Element method
*3)* Finite Volume method

### *D. Finite Volume Method*

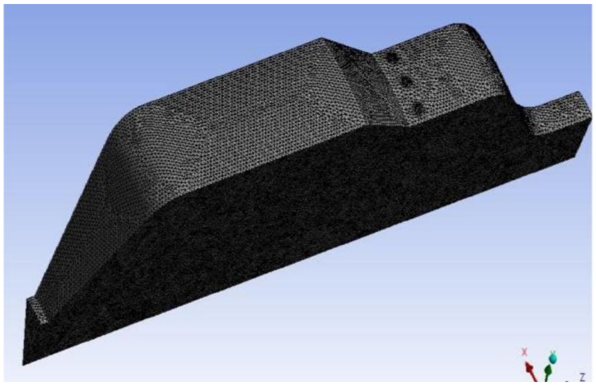
Finite volume method was actually derived from finite element method. The numerical algorithm for these methods consists of following steps,

*1)* Integration of the governing equations of flow over all the finite control volumes of the computational domain. This step distinguishes this method from the other discretization techniques.
*2)* With some finite difference approximations these integral equations are being algebraic equations.converted into the
*3)* These algebraic equations are solved on an iterative basis One such commercial CFD code, which uses the finite volume approach, is FLUENT, which has been employed for the present study

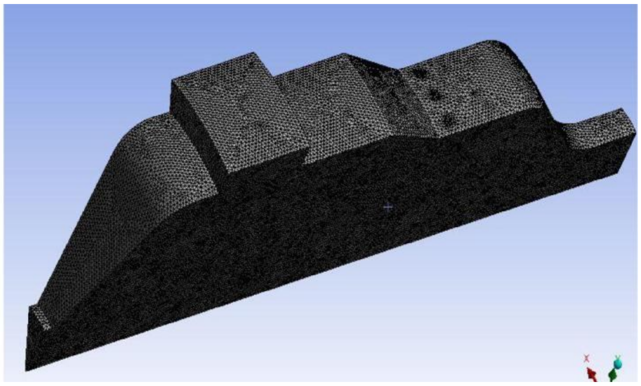
### *E. Quality Of The Grid Generated*

As the accuracy of the any CFD solution depends on the quality of the grid used, we have checked the quality of the grid using two parameters via, Skewness cells, which have skewness in the region of 0.3 to 0.85.For structured and in the region of 0.3 to 0.9. For unstructured, are considered as good quality cells. For the we have generated the maximum equi-angular skewness we have found to be 0.72 , which indicated that the grid is of a good quality .Aspect ratio for a rectangular face is the length to breadth ratio of it. For a good quality cell it ranges from 1to10. For the grid we generated the maximum aspect ratio is 6.5.

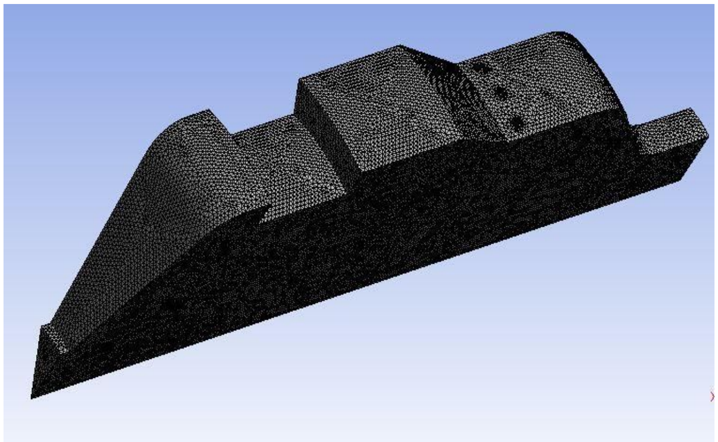
Type of Mesh
Unstructured Mesh
Type of Element
Mixed (Quad & Tri)
Quality of Mesh-
Skewness (0.59)
No of Elements -
1, 26,072 (Normal combustor)
1, 43,230 (Model 2)
1, 59,933 (Model 3)



Meshed normal combustor



meshed trapped vortex combustor model 1



Meshed trapped vortex combustor model 2

*F. Boundary Conditions*

Table Inlet Boundary Conditions

| Condition | Primary | Secondary |
|---|---|---|
| Flow Regime | Subsonic | Subsonic |
| Mass Flow Rate | 0.25 kg/s | 0.29 kg/s |
| Total Temperature | 575K | 575K |
| Flow Direction | r Component = 0<br>Theta Component = 0.866<br>Axial Component = 0.5 | Normal to Boundary |
| Fuel Mass Fraction | 0.038 | --- |
| O2 Mass Fraction | 0.223 | 0.223 |

Table .Outlet Boundary Conditions

| Condition | Primary |
|---|---|
| Flow Regime | Subsonic |
| Pressure Outlet | 732 kPa |
| Temperature | Local Temperature |

## V. RESULTS AND DISCUSSION

### A. *Normal Combustor Model Results*

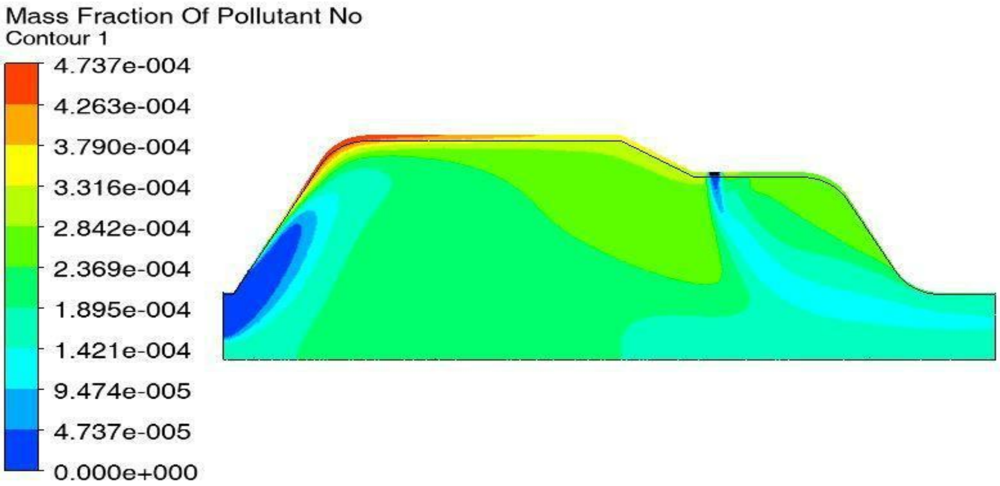


Fig Contours of Pollutant $NO_X$ Emission

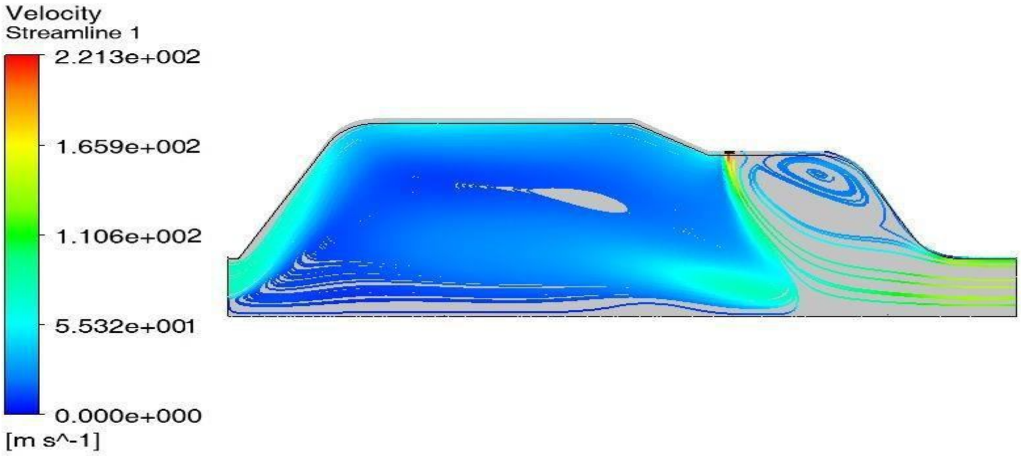


Fig Path lines showing flow pattern inside normal combustor

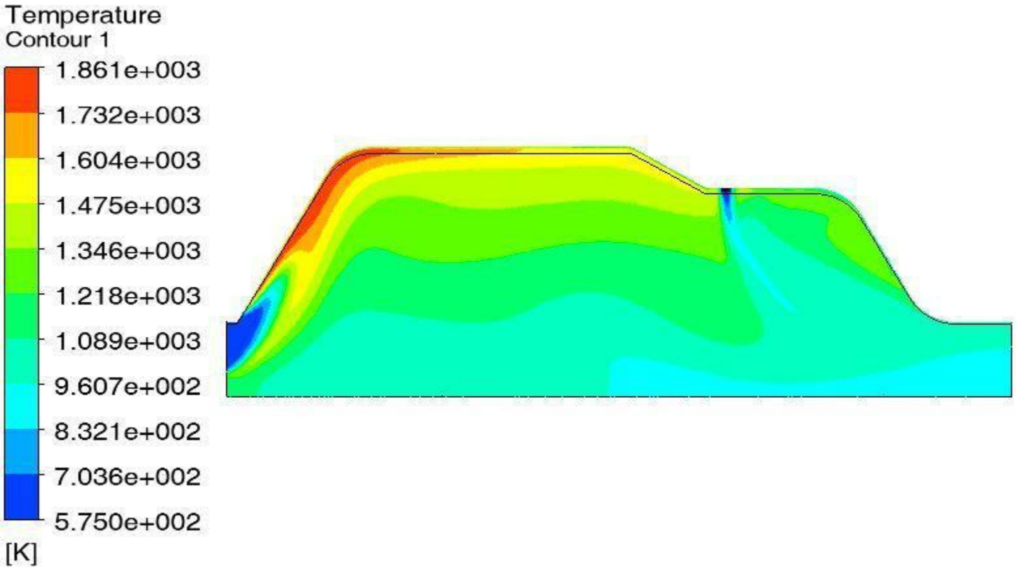


Fig Contours of temperature in normal combustor

*B. Trapped Vortex Combustor (Model 2) Results*

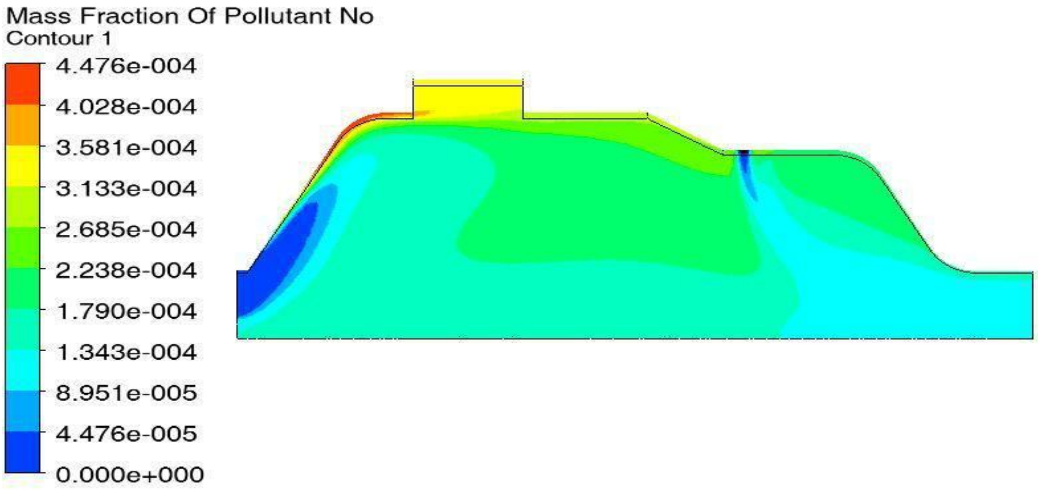


Fig Contours of Pollutant $NO_X$ Emission

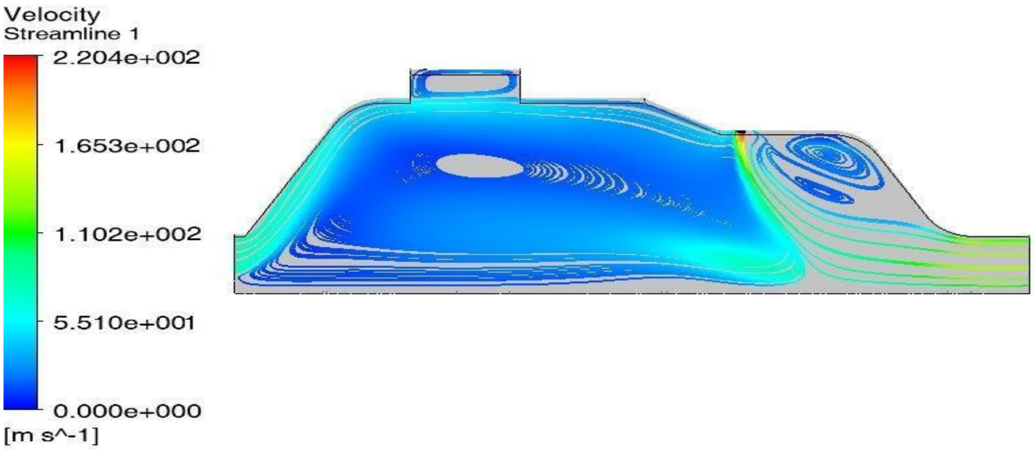


Fig Path lines showing flow pattern inside Model 2

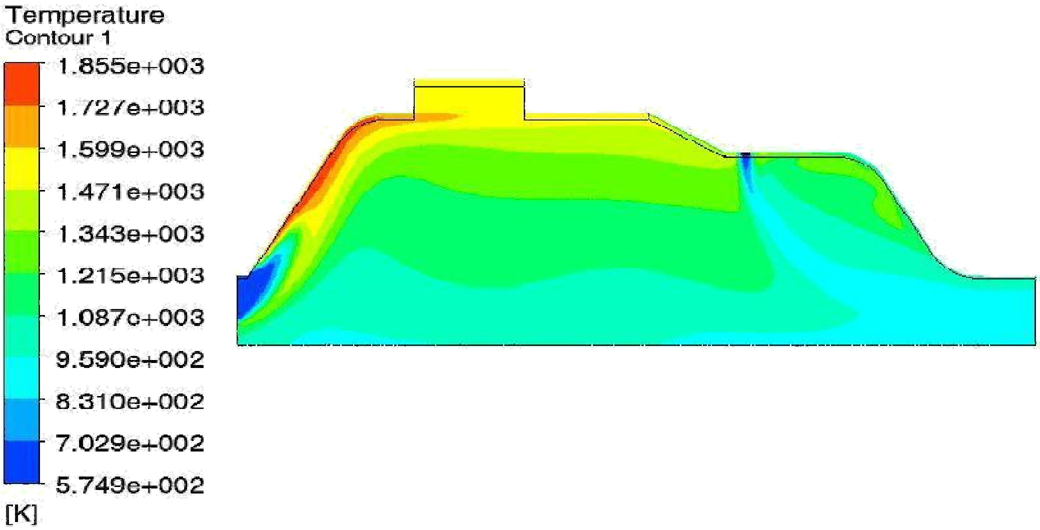


Fig Contours of temperature in Model 2 Combustor

### C. *Trapped Vortex Combustor (Model 3) Results:*

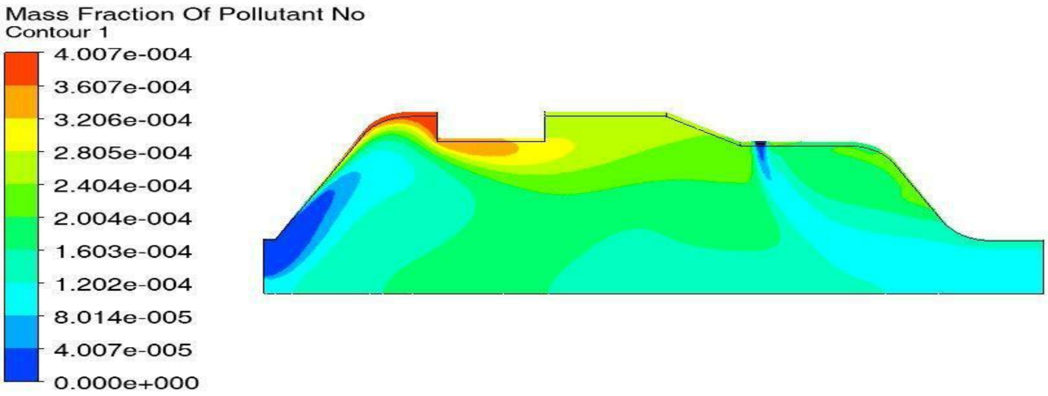


Fig Contours of Pollutant $NO_X$ Emission

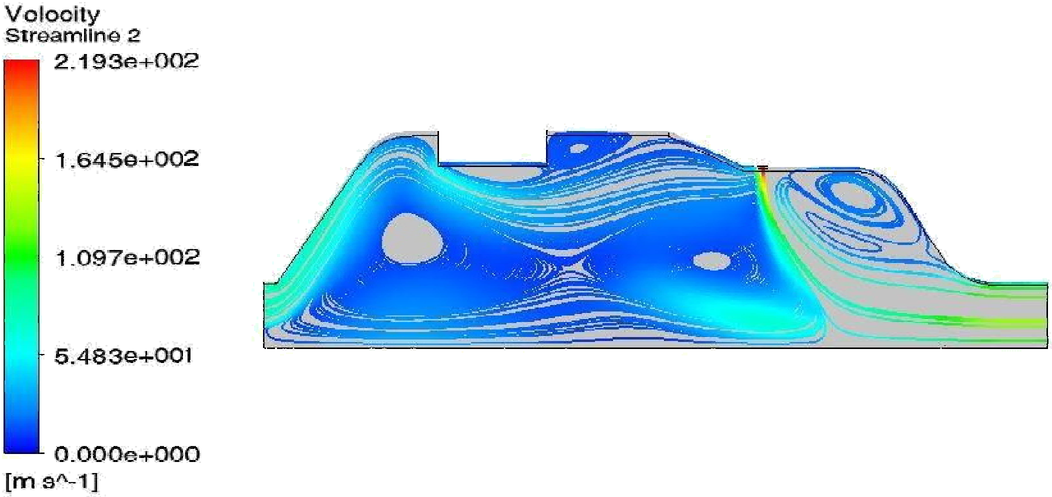


Fig Path lines showing flow pattern inside Model 3

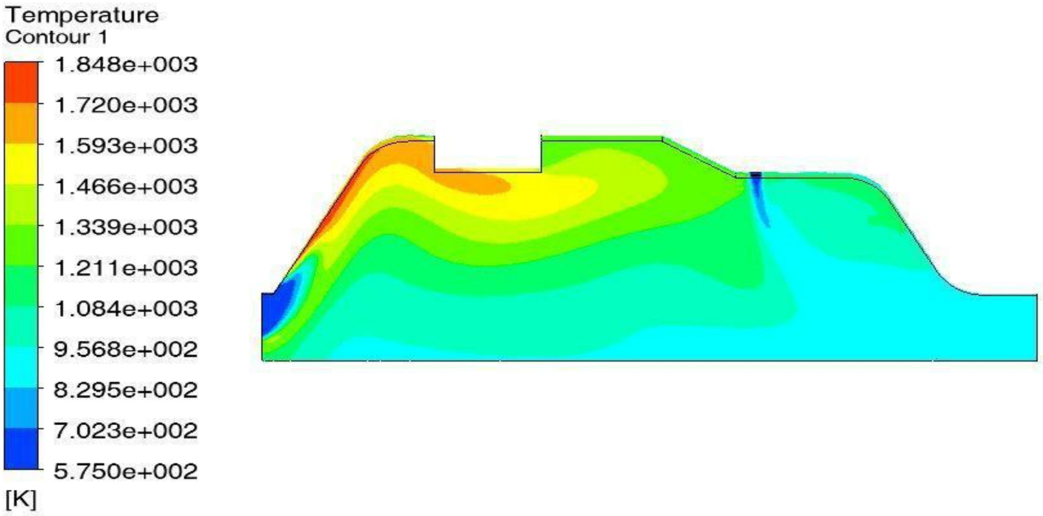


Fig Contours of temperature in Model 3 Combustor

### D. *Results Summary*

Table Results Summary of Present Work

| | Normal | Vortex 1 | Vortex 2 |
|---|---|---|---|
| Maximum temperature (K) | 1860 | 1855 | 1848 |
| NO emission (ppm) | 473 | 447 | 400 |
| Outlet Pressure (bar) | 3.15 | 3.08 | 2.85 |

Table shows the results summary of present work. From the table it seems that trapped vortex combustor model with inside block has less emission compared to other two models.

Validation Results Summary

| | Preliminary design | CFX | Fluent (Present work) | Error (%) |
|---|---|---|---|---|
| Maximum temperature (K) | 1803 | 1885 | 1860 | 3 |
| Turbine Inlet temperature (K) | 1203 | 1190 | 1158 | 4 |

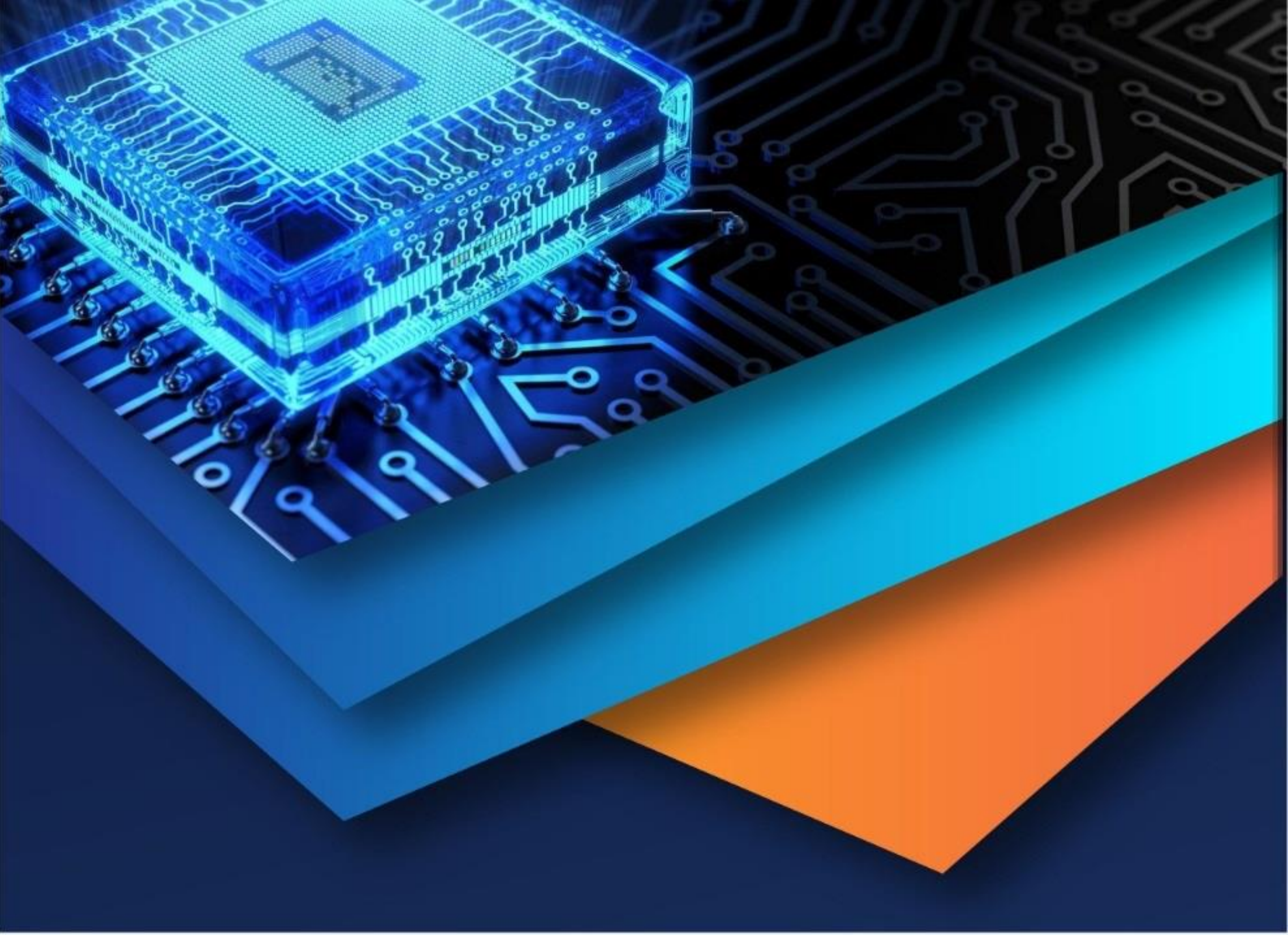
## VI. CONCLUSION

### A. *Conclusion*

A combustor model was modeled and emission characteristics was investigated by numerical simulation. Numerical simulation of the combustor model was validated with Preliminary design and existing CFD analysis values. The maximum flame temperature and turbine inlet temperature was validated and the error was found to be 4 % which is marginable. Two different trapped vortex combustor configuration was proposed for the existing normal combustor and its performance was investigated numerically. From the results summary, it leads to conclusion that TVC enhance flow mixing by increasing the flow recirculation in the primary zone. 20% emission reduction was achieved by implementing TVC in normal combustor.

## REFERENCES

[1] Anderson, W.S., Radtke, J.T., King, P.I., Thornburg, H., Zelina, J., Sekar, B., “ Effects of Main Swirl Direction on High-g Combustion,” 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 21-23 July 2008, Hartford, CT., AIAA2008-4954, 2008.
[2] Anderson, W., Bonhus, D.L., “An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids,” Computers Fluids, 23(1):1–21, 1994.
[3] ANSYS FLUENT 12.0, Theory Guide, 2009.
[4] Brushwood J (1999) Syngas Combustor for Fluidized Bed Applications 15th Annual Fluidized Bed Conference.
[5] Briones, A.M., Sekar, B, ., Thornburg, H., Zelina, J., “Effect of Vane Notch and Ramp Design on the Performance of a Rectangular Inter-Turbine Burner,” 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, Orlando, FL, AIAA-2010-581.
[6] Blomeyer M, Krautkremer B, Hennecke DK and Doerr T (1999) Mixing Zone Optimization of a Rich-Burn/Quick-Mix/Lean-Burn Combustor. Journal of Propulsion and Power 15: 288-303.
[7] Barth, T. J., Jespersen, D. “The Design and Application of Upwind Schemes on Unstructured Meshes,” AIAA-89-0366, 27thAerospace Sciences Meeting, Reno, Nevada, 1989.
[8] Bird, R.B., Stewart, W.E., and Lightfoot, E.N., Transport Phenomena, Wiley, New York, 1960.
[9] Cline MC, Micklow GJ, Yang SL and Nguyen HL (1992) Numerical Analysis of the Flow Fields in a Rql Gas Turbine Combustor.
[10] Feitelberg AS, Jackson MR, Lacey MA, Manning KS and Ritter AM (1996) Design and Performance of a Low Btu Fuel Rich-Quench-Lean Gas Turbine Combustor. Advanced coal-fired power systems review meeting. USA DOE, Morgantown Energy Technology Center.
[11] Feitelberg AS and Lacey MA (1998) The Ge Rich-Quench-Lean Gas Turbine Combustor. Journal of Engineering for Gas Turbines and Power, Transactions of the ASME.120: 502-508.



10.22214/IJRASET

INDEX COPERNICUS
45.98

ISRA JIF
IMPACT FACTOR:
7.129

SJIFactor
TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)