International Journal of Architecture, Engineering and Construction ISSN (online) 1911-1118

Vol 11, No 1, March 2022 (UGC CARE 1) ISSN (print): 1911-110X

COMPUTATIONAL AND EXPERIMENTAL INVESTIGATION OF FLOW AND COMBUSTION CHARACTERISTICS IN A CAN-TYPE COMBUSTOR

Dr. K. Kalyani Radha

Assistant Professor, *Dept. of Mech Engg, JNTUCEA, Ananthapuramu, A.P, INDIA.*

Abstract: **The main aim of this work is to study the flow field of can-combustor as preliminary approach for gas turbine combustor application. Can-combustor geometry is modeled using solid works, meshing and analysis is carried out using Ansys workbench. Mass flow distribution through the various zones of the Can-combustor has been calculated by cold flow CFD analysis. Can-combustor contains various components like swirler, dome, primary holes, dilution holes andcooling holes. Total pressure loss through each componenet decides the mass flow distribution in the various zones of combustor. Hot flow CFD analysis of Can-combustor has been carried out with propane gas as fuel. It shows the presence high temperature region in the central region of the liner. This combustor is designed for an overall equivalence ratio of 0.125 for preheating applications with the exit temperature of 650ºK. In hot flow analysis fuel intake in to the can-combustor is varied for wide range of equivalence ratio (0.125 to 0.5), considering air inlet a constant value. The hot flow analysis depicts a clear picture regarding temperature distribution in the can-combustor. The distribution of recirculation region is estimated with the flow analysis. While species transport without combustion made us to evaluate the effect of mixing and the formation of stoichiometric equivalence ratio.**

Keywords: Can-combustor, CFD simulation, Non premixed combustion, species transport.

1. INTRODUCTION

The main aim of combustion system is to convert chemical energy hydro carbon (fuel) compounds injected in to the system to thermal energy of the system and get uniform temperature profile at exit. A complete combustion consists of water vapor and carbon dioxide at the exit. A combustor should hold stable flame for long period of time for different loading conditions of pressure and equivalence ratios, otherwise it would result in the lean blow out which is undesirable for the combustion system. Most recent engines use Can-Combustors are turbo shaft featuring centrifugal compressor. They are self-contained combustion chambers and cylinder from origin. The compressed air is divided and guided to each discrete can, where the magnitude of velocity gets decreased for mixing with fuel and then ignition will start. The secondary air comes from the compressor is given outside the liner and forms a sheet and in core the combustion will be taking place. These can-combustor tubes are interconnected. The major contribution of the can-combustor is they are arranged circumferentially around the axis of the engine and their outlet is given to the turbine inlet.The Advantages of can combustor is ease of testing and design. Ease of maintenance and replacement, if one combustor get affected it can be replaced without affecting whole system.Individual can-combustor is used for small and auxiliary power units.The challenges come across while using can combustors are Pressure losses up to (7%) of the total inlet pressure.The overall pressure loss in a gas turbine combustor originates mainly due to losses in the diffuser while conversion of high velocity gases (kinetic energy) to pressure energy the significant losses is due to fluid flow separation and formation of the eddies at the enlarged section in turn local pressuregradient acting on the incoming molecules and also friction acting on these particles by wall which will be discussed later.Mixing of incoming main stream flow with the radial jet holes. There is a significant loss of pressure during this process. While combustion due to phase transformation and exothermic interactions of molecules with the compressed air.Losses due to wall friction. The nature of frictional losses are generally considered as linear variation along the length of the combustor.

Weight of the system increases with the increase in the number of cans compared to conventional systems. Ignition problems come at higher altitudes. The main aim of this work is to know the flame propagation region in premixed combustion and species interaction using CFD.

2.LITERATURE SURVEY

Methuselah G (2012)[1] has done a computational Study of recirculating flows Induced by axial Swirler. Mass flow rate through the swirler reduces with increase in swirl angle due to increase in resistance to flow. The length of CTRZ increases with swirl angle initially, but after 50^0 it is almost maintained constant. Audai Hussein (2011)[2]has

performed CFD modeling of air-fired and oxy-fuel combustion in a 100 kW unit firing propane 3-D hybrid unstructured grid CFD code. It uses a swirl injection system to achieve the flame stability of the turbulent non-premixed combustible gases. The swirl effect is certainly used to enhance the turbulent mixing and to achieve the internal recirculation of flames. The residence time of combustible reactants is considerably increased due to adopting the swirl injection system that leads to enhance flame stability.

H. M. Heravi (2011) [3] with use of swirl, flow region is divided to the central toroidal recirculation zone and corner recirculation zone. The maximum flame temperature tends to increase with the inlet conditions of combustor as the swirl number increases. ZHANG Xin(2011)[4]did a 2D numerical simulation on the laminar non-premixed combustion of n-heptane and air. Fuel-air equivalence ratio has a big influence on the characteristics of non-premixed Combustion. Inlet gas velocity strongly influences the combustion characteristic, such as combustion zone and flame plume length, while it has no effect on the flame temperature. ChaoukiGhenai(2010)[5] investigated the Combustion of Syngas fuel in Gas turbine can-combustor. The results of the gas temperature, velocity field, swirling strength and CO_2 and NO_Xemissions show that gas turbine can-combustor burns the fuel efficiently, reduces the emissions and lower the wall temperature.

The predicted maximum temperature of methane fuel combustion compares well with the theoretical adiabatic flame temperature. Yehia A. (2010) [6]has performed Cold flow investigation of primary Zone characteristics in combustor utilizing axial Air swirler. Air swirler is one of the most effective ways to induce flow recirculation inside the primary zone. This type of recirculation provides better mixing. In addition swirling flow is used to control the stability and intensity of the combustion and the size and shape of the flame region which is dependent on the size and shape of the recirculation zone. C.E.L. Pinho (2008) [7]has done numerical study of propane-air mixture combustion in a burner element. The results show that the increase of equivalence ratio corresponds to a significantly decrease in the maximum reaction rates The maximum temperature increases with the increase of oxygen percentage.

Jian Zhang (2007) [8]has done a numerical simulation of turbulent reacting flow in a swirl combustor. The gas temperature and carbon dioxide concentration are high in the central region and relatively low in the near wall region of the combustor. While the propane and oxygen concentrations are low in the central region and high near the combustor wall. The distributions of gas temperature and species concentration tend to be uniform at the combustor exit.Guoqiang Li (2006) [9]have studied boundary condition effects on non-reacting and reacting Flows in a multiswirl Combustor. This experimental study emphasized the significant impact of inlet and outlet boundary conditions on the non-reacting and reacting flows.

Increased air inlet temperature enhanced the magnitude of reversed flow in the central region and increased the tangential velocity in the swirling jets and the energy of the turbulent velocity fluctuations.

S. Bharat Krishna and V.Ganesan (2005) [10] investigated about a computational study of steady flow through vane swirler for various vane angle of 15º to 60ºin steps of 15ºand turbulence has been modeled using Reynolds Stress Model (RSM).They also measure variation in the velocity downstream of swirler for various swirl vane angles and pressure variation in the flow field.Fernando F (2002) [11] in flow dynamics in a swirl investigated the flow patterns in an axisymmetric swirl combustor configuration. The flow is driven by the strong interaction between swirling shear layer instabilities, on the one hand, and flow instabilities driven by the sudden expansion and geometry of the combustion chamber to other side. M.D. Durbin (1996) [12]studied the Lean Blowout in a Step Swirl Combustor. They used a rectangular chamber which has fuel supplied through annular fuel tube which is coaxially sandwiched between two swirling air streams. They concluded that Flame length studies for different vane angles Co swirl vs. counter swirl effects on the hot region and temperature distribution. Marc R.J. Charest (1995)[13],author had investigated on the preliminary requirements of a can-combustor and validated obtained model using CFD. They had predicted the CTRZ in the primary zone and shown the importance of jet mixing. In hot flow of their premixed combustion they had predicted that the high temperature lies near the liner as the central recirculation region entrains the cold air and mixes with the flow and thus reduces the temperature.

3. CFD PROCEDURE AND CONDITIONS

Fig.1 shows cross sectional view of can combustor

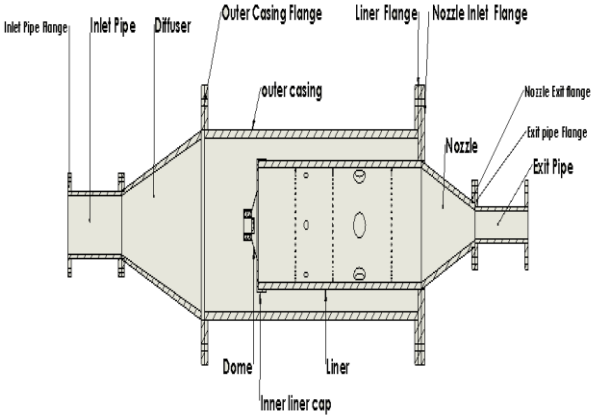


Figure.1 Cross sectional view of the designed can-combustor

considered for this present CFD analysis. Geometry of this can combustor consists of diffuser, casing, liner, swirler, dome and nozzle. Fuel injector was assembled with axial swirler. The geometry consists of one row of primary and dilution holes. Three rows of liner cooling holes and one row dome cooling holes. This can combustor was modeled using solid work software. Fig.2 shows computational domain used in the present CFD analysis. To calculate mass flow rate through each zone of combustor several interior planes were created as shown in figure.2. Unstructured tetra mesh was created using ANSYS meshing software as shown in fig.3.To effectively capture temperature gradients downstream of swirler fine mesh is generated in the primary zone of combustor. Coarse mesh is generated in the stagnation zone of the can-combustor. Very fine mesh is generated near dome cooling holes and liner cooling holes.

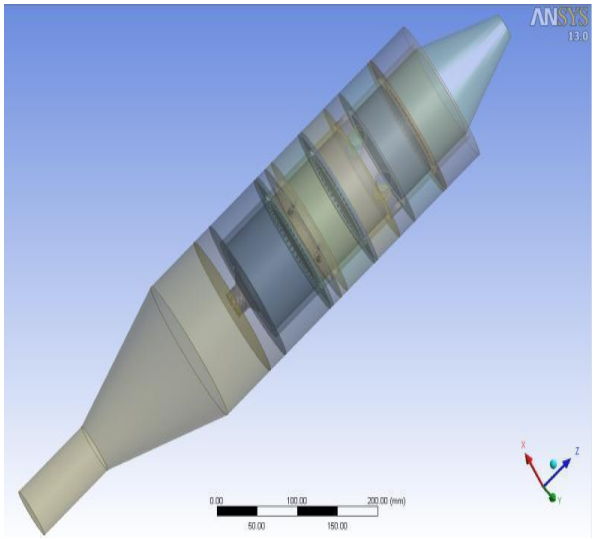


Figure.2 CFD model domain designed for can-combustor

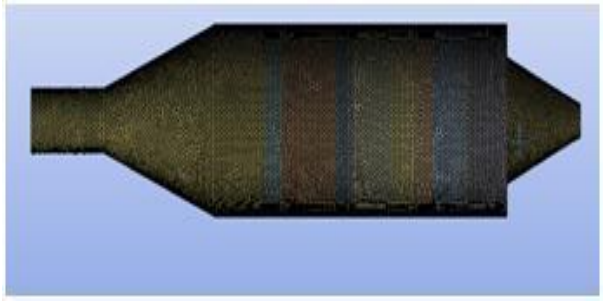


Figure.3 CFD meshing domain designed for can-combustor

BOUNDARY CONDITIONS

BOUNDARY CONDITIONS FOR COLD FLOW

Air Inlet: Pressure inlet

Outlet: Pressure outlet

Turbulence model: K-ε model (standard)

Reference pressure: 91700 Pa

Parameters	Values
Inlet pressurePa	3000
Inlet Temperature º K	300
Air mass flow rate(kg/sec)	0.0414
Exit pressurePa	0
Exit Temperature º K	300

BOUNDARY CONDITIONS FOR HOT FLOW:

Air Inlet: Pressure inlet

Fuel inlet: Mass flow inlet

Outlet: Pressure outlet

Turbulence model: K-ε model (standard)

Reference pressure: 91700 Pa

Fuel: Propane

Parameters	Values
Inlet PressurePa	5500,8200,11200,12500,14500
Inlet TemperatureK	300
Airmass flow rate (kg/sec)	0.0414
Fuel flow rate (kg/sec)	0.00032,0.00053,0.00079,0.0010,0.0013
Exitpressure Pa	0
Exit Temperature(°K)	652.06,843.28,1081.51,1302,1507.4

NOMENCLATURE

Ø– Equivalence Ratio

Lch 1 - Liner cooling holes 1

Pz - Primary zone

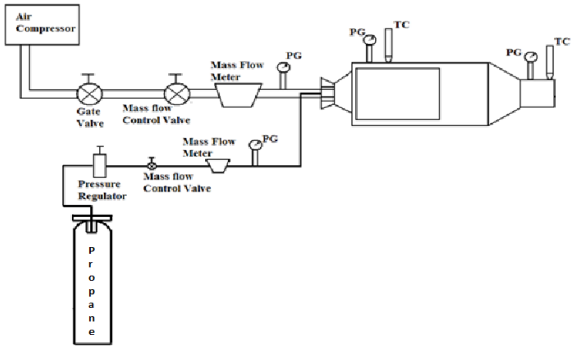
Lch 2 - Liner cooling holes 2

Dz 2 -Dilution zone

Lch 3-Liner cooling holes 3

4.EXPERIMENTAL SETUP

Fig.4 shows schematic diagram of experimental setup used in testing of can-combustor. In this experimental setup both air and fuel pressure can be independently controlled, to simulate different equivalence ratios. Coriolis mass flow meters were used to accurately measure the mass flow rates of fuel and air, which is shown in the fig.5. Six different thermocouples are used to measure temperature at different locations of can-combustor during hot flow experiment. The combustor with thermocouples is shown in the fig.6. Three thermocouples are used at the exit of the combustor to measure combustor exit temperature at different equivalence ratios. Fig.7 schematically shows the placement of thermocouples at different axial locations of can-combustor.



Flow by each zone	Percentage of mass flow rate
SWIRLER	5.094
DOME COOLING HOLES	5.324
LCH1	5.423
PRIMARY ZONE	10.36
LCH2	5.5818
DILUTION ZONE	62.882
LCH3	6.1
TOTAL	100

Table.1Percentage of mass flow distribution in cold flow for 60°swirler

Figure.4 schematic diagram of experimental setup



Figure.5 Experimental setup of can-combustor testing

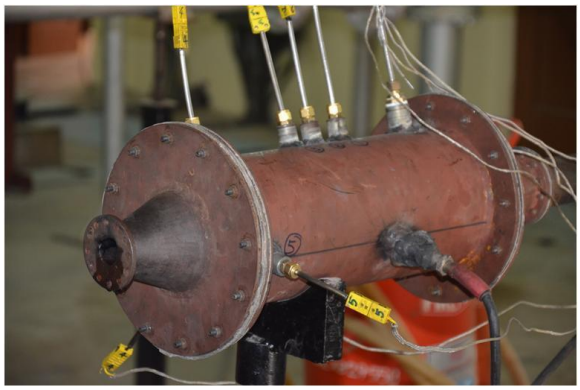


Figure.6 Can-combustor with six thermocouples

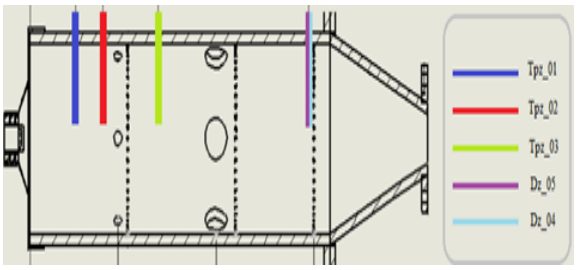


Figure.7 Location of thermocouples used to measure temperature in the can-combustor

5. RESULTS AND DISCUSSIONS

Cold flow CFD results

Fig. 8 shows cold flow midplane velocity contour plot of can-combustor and Fig.9 depicts the direction of velocity vector. CFD analysis clearly predicts the central toroidal recirculation zone of swirler and vortices produced due to dilution jets inside the liner. From the cold flow CFD analysis, mass flow distribution through various zones of can-combustor has been calculated and shown in table1.

Velocity vectors during the cross flow interactions create a counter rotating vortices. The interaction of jets downstream of swirler inlet gets dissipated this can be clearly visible from Fig.10 which is taken at 104mm downstream from fuel inlet**.** In the later case Fig.11. 130mm down the stream from swirler inlet the intensity of the jets decreases due to interaction.

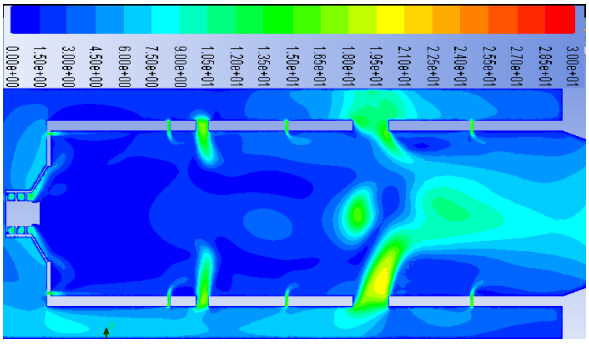


Figure.8 Mid plane Velocity contours of can-combustor

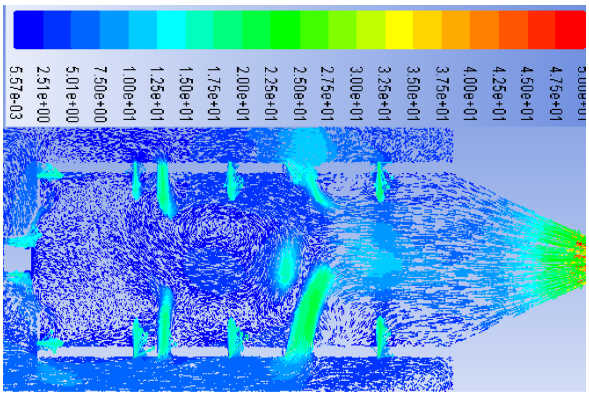


Figure.9 Mid plane Velocity vectors of can–combustor

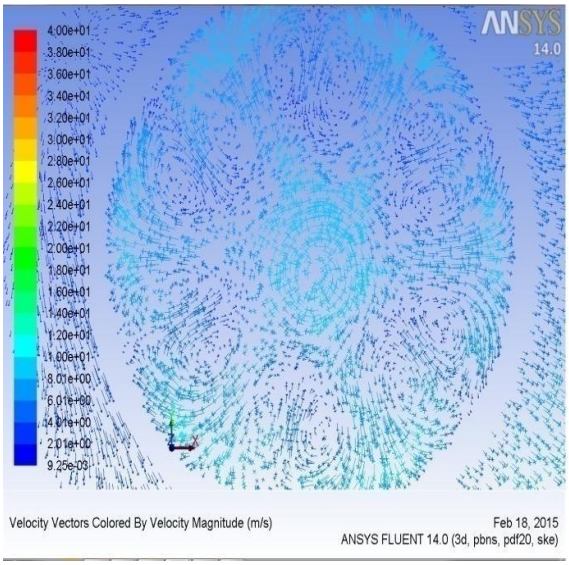


Figure.10.Velocity vectors 104mm down the stream from fuel inlet

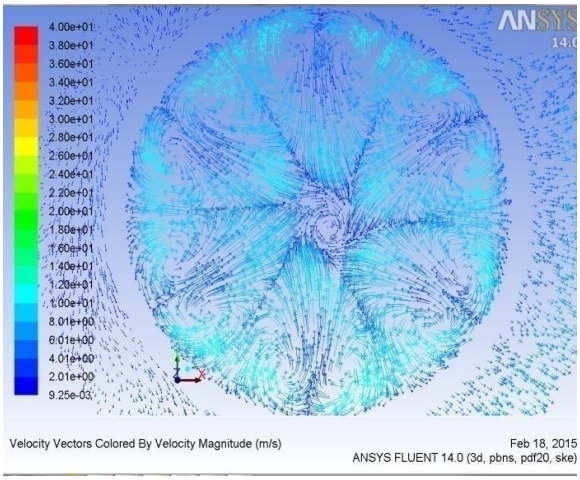


Figure.11.130mm velocity vectors down the stream from fuel inlet

Hot flow CFD Results

Hot flow CFD analysis has been carried out with propane gas as fuel. Fig.12 shows midplane temperature contour of can-combustor. This can-combustor geometry contains three rows of linear cooling holes and one row of dome cooling holes. These

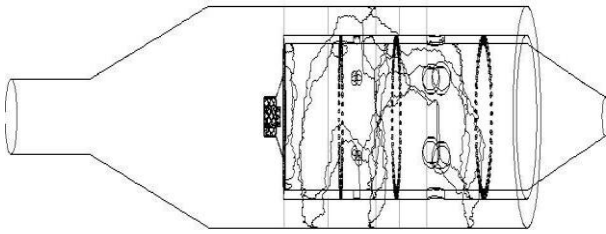


Figure.13 Presence of radial planes in the combustor for temperature analysis

cooling holes allow cold air into the hot regions of combustor and cool the liner effectively as shown in the Fig.12. At the top and bottom edges of dome region, high temperature zones are present due to the formation of corner recirculation zones. From Fig.12, it can be concluded that, most of the combustion happens in the primary zone due to availability of excess oxygen through the swirler and primary holes.

High temperature region is well placed in the central region of the liner. Fig.12 shows temperature contour at different axial locations of can combustor. Entry of air through different rows of cooling holes, and mixing of primary and dilution jets with central core flow can be seen in the Fig.12. In table.2, exit parameters of can combustor are compared with results of NASA CEA code.Several axial planes were created in the combustor as shown in the Fig.13and mass weighted average temperatures were taken and plotted in Graph.2

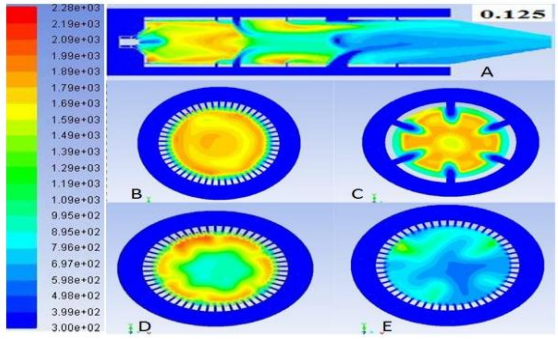
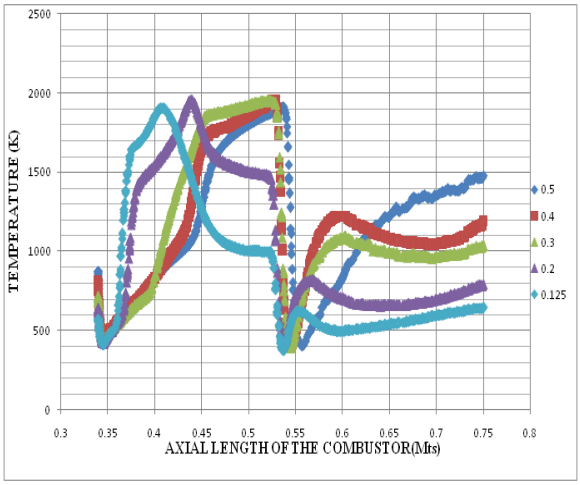


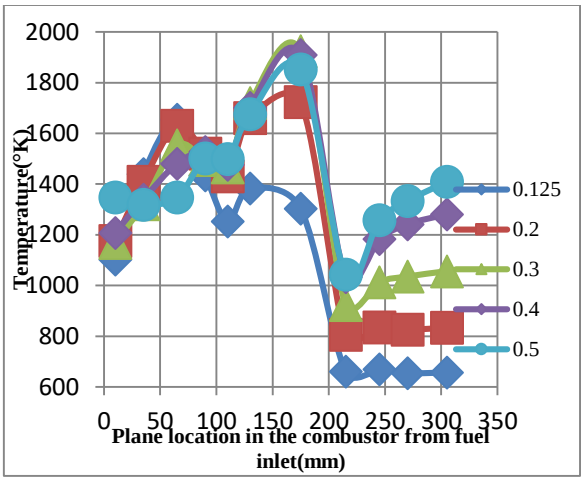
Figure.12 Hot flow midplane temperature contour of can combustor and at different axial locations of can combustor for an overall equivalence ratio of 0.125

Table.2 Comparison of exit parameters between NASA

Parameters	NASA CEA code	CFD Results
Temperature (K)	651.48	648.16
Mass fraction of CO_2	0.0242	0.023
Mass fraction of H_2O	0.013	0.013
Mass fraction of N_2	0.749	0.761
Mass fraction of O_2	0.2	0.2

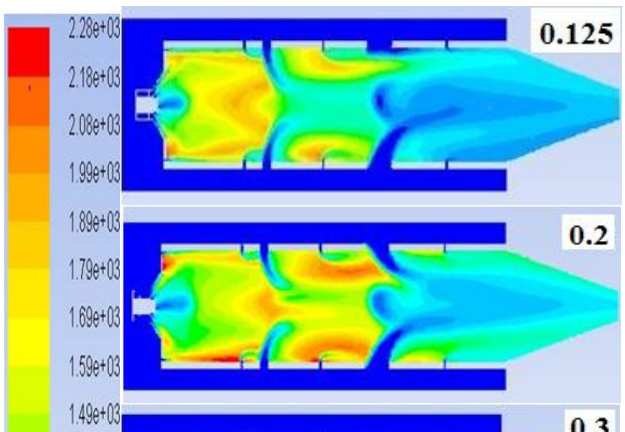


Graph.1Variation of temperature along the axial line from fuel inlet to the exit for different equivalence ratios (ϕ)



Graph.2 Variation of temperature at different radial planes from fuel inlet with increase in equivalence ratio

Axial line is drawn is along the axis of the combustor from fuel inlet to the nozzle exit and temperatures (^oK) were plotted in Graph.1. At lower equivalence ratios combustion is taking place in the primary zone only and peak value of temperature is reached. With increase inequivalence ratios the flame temperature shifts towards the dilution zone and can be easily visible frommid plane temperature contour of figure16



Hot flow experimental results

Figure.16 Mid plane Temperature (K) contours of can-combustor representing different equivalence ratios such as 0.125, 0.2, 0.3, 0.4, 0.5 on a common scale.

OVERLAY OF SPECIES TRANSPORT POINTS AND TEMPERATURE CONTOUR OF NON PRE MIXED COMBUSTION

At lower equivalence ratio 0.125 the entire stoichiometric equivalence ratio points lies within the primary zone and with the increase in the equivalence ratio the points shifts towards the dilution holes. A similar observation is observed in the fig.17 which represents stoichiometric equivalence ratio points on mid plane of the combustor. These were found out by using species transport without combustion only mixing will be taking place in the combustor flow field. Overlay of species transport points and temperature contour of non pre mixed combustiongives a reasonable stoichiometric mixing in the primary zone at lower equivalence ratio as shown in figure17. When equivalence ratio increases, the fuel inlet velocity increases and in stoichiometric mixing there is a shift which is valid with the combustion chemistry.

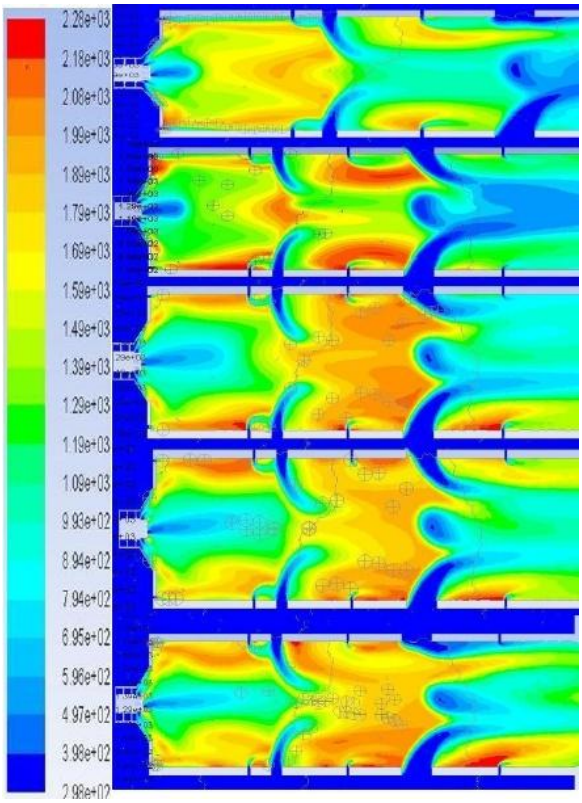
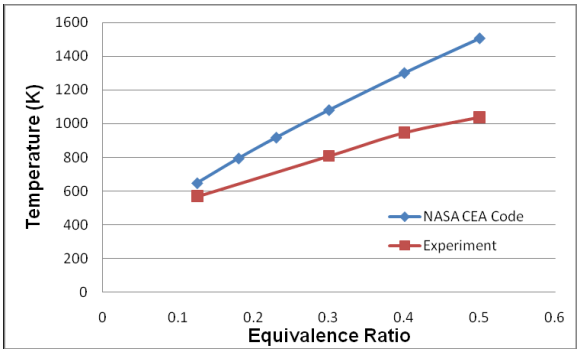


Figure.17 overlay of species transport points on temperature contour of non pre mixed combustion

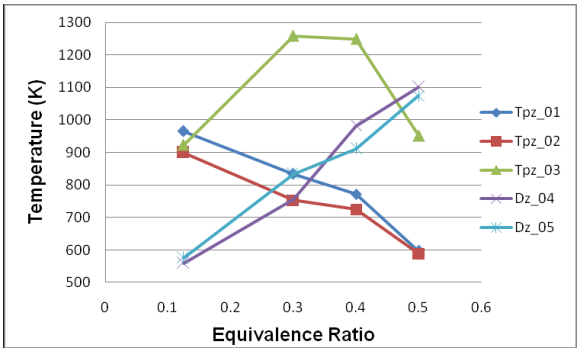
Hot flow experimental results

In Graph.3 measured values of temperature at exit of the can-combustor for various values of equivalence ratios compared with the results of NASA CEA code. In both curves, temperature increases with increase in overall equivalence ratio. But at higher equivalence ratios, experimental values deviate from the theoretical results, due to higher heat loss through wall of combustion chamber. Graph.4 shows the measured values of temperature at different locations of can-combustor for various values of equivalence ratios. Both Tpz-01 and Tpz-02 are located ahead of primary holes. Tpz-03 located between primary and *dilution* holes. Tdz-04 and Tdz-05 are located downstream of dilution holes. Propane gas is injected through 11 holes (8 radial and 3 axial) of 2mm diameter. After successful ignition of fuel air mixture, mass flow rate of fuel continuously increased (with constant air flow rate) to reach overall equivalence ratio of 0.5 and reduced till flame extinction. Up to the equivalence ratio of 0.2, maximum temperature was observed in the primary zone. At this condition of fuel flow rate, the air flow available in the primary zone is enough to completely burn the fuel. Further

increase in equivalence ratio (increase in fuel flow rate), shifts the maximum temperature region downstream of the primary zone as shown in Graph.4, due to unavailability of air in the primary zone for complete combustion offuel. At the overall equivalence ratio of 0.5 the maximum temperature zone has been further shifted and observed near dilution zone, which can be seen in the Graph.3, Dz_04, Dz_05 reads higher temperature than primary zone temperature



Graph.3comparison of exit temperature at different equivalence ratios with results of NASA CEA



Graph.4 Temperature at different axial locations of can combustor at different equivalence ratios

6. CONCLUSION

From the above observations it is noticed that the results ofNASACEA code, results of non-premixed combustion module and species transport in Ansys fluent were compared and it is found good agreement with respect to mass fractions.

CFD COLD FLOW ANALYSIS:

- Pressure losses increases with the increase In swirler vane angle.
- Can-combustor flow field has been predicted and mass flow distribution through each zone is computed.

CFD HOT FLOW ANALYSIS USING NON PRE MIXED COMBUSTION

- Temperature contour at different axial distance has been compared and temperature distribution along combustor has been understood.
- With increase in equivalence ratio the maximum flame temperature shifts towards the dilution zone.
- At the design point of an overall equivalence ratio of 0.125, required exit temperature of 650K has been achieved in can-combustor. Further increase in equivalence ratio shifts the maximum temperature region downstream of the primary zone due to unavailability of air in the primary zone for complete combustion of fuel

REFERENCES

[1].Muthuselvan-2012, Knight, H. A.; and Walker, *The Component Pressure Losses in Combustion Chambers. ARC R&M 2987, 1957.*

[2] Hussein Mixing of multiple gas jets subjected to a cross flowLeonard s cohen and William j Egan

[3].Heravi F.S Billig, R.C Orath and M.Lasky *A Unified Analysis of Gaseous jet penetration*

[4]. *Xin Losses in Combustion Chambers. 1957.*

[5]. ChaoukiGhenai- "Combustion of Syngas Fuel in Gas Turbine Can-combustor"- Advances in Mechanical Engineering-2010

[6]. Yehia A. Eldrainy, Mohammad Nazri Mohd. Jaafar, Tholudin Mat Lazim, "Cold Flow Investigation of Primary Zone Characteristics in Combustor Utilizing Axial Air Swirler*" International Journal of Mechanical and Materials Engineering 2010*

[7].Pinho Jack D. Mattingly, *William H. Heiser, David T. Pratt "Aircraft Engine Design"AIAA*

[8]. Zhong Beijing, Wu Heng "Numerical simulation on catalytic combustion of CH4/air in micro-channel" *Journal of Combustion Science and Technology, 2005 (2): 1-5*

[9]. Guoqiang Li and Ephraim J. Gutmark – "Boundary Condition Effects on Nonreacting and Reacting Flows in a Multiswirl Combustor"- University of Cincinnati,Cincinnati, *AIAA J OURNAL Vol. 44, No. 3, March 2006*

[10].V.ganeshan HuaJinsong, Wu Meng, Kurichi K "Numerical simulation of the combustion of hydrogen-air mixture in micro-scaled chambers (II): CFD analysis for a micro-combustor" *Chemical Engineering Science, 2005, 60 (13): 3507-3515.*

[11]. Fernado S. Kamnis, S. Gu "Numerical modelling of propane combustion in a high velocity oxygen–fuel thermal spray gun" *Chemical Engineering and Processing 45 (2006) 246–253.*

[12]. M.D.Durbin,D.R. Ballal "Studies of Lean Blowout in a Step Swirl Combustor"*ASME,vol 118,Jan1996*

[13]. Charest JIANG Liqiao, ZHAO Daiqing, and WANG Xiaohan "*Structure and extinction characteristics of methane micro-diffusion flames" 2007, 13(2): 183-186*