International Journal of Scientific Research in Science, Engineering and Technology
© 2017 IJSRSET | Volume 3 | Issue 5 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
doi : https://doi.org/10.32628/IJSRSET1735145

Numerical Investigation on Variation of Axial and Radial Velocity Profile Inside CAN Type Combustion Chamber

Prakashkumar R. Patel[2], **Ambaliya Sanjaykumar D**[2]
[1]Assistant Professor, Mechanical Engineering Department, Dr. S. & S. S. Ghandhy Government Engineering College, Surat, Gujarat, India
[2]Assistant Professor, Mechanical Engineering Department, Government Engineering College, Valsad, Gujarat, India

ARTICLEINFO

Article History:
Received : 07 June 2017
Revised : 05 August 2017
Accepted : 25 August 2017

Publication Issue :
Volume 3, Issue 5
July-August-2017
Page Number :
831-838

ABSTRACT

Two dimensional numerical simulation of reactive flow in coaxial can type combustion chamber is performed. Percentage in air is varied to investigate its effect on combustion combustor flow parameters. The effect of oxygen percentage in the combustion air are investigated fir value of equivalence ratio, and from 10% to 30%. Simulation is performed for constant energy supply to combustor (Q). Preprocessing is carried out using GAMBIT as a preprocessor. Commercially available code FLUENT is used as a solver. Results are compared and observed that combustion reaction rate gets enhanced with increased percentage of oxygen.

Keywords : Combustor, Numerical Simulation, Velocity Profile, Oxygen Percentage

I. INTRODUCTION

1.1 Gas Turbine Combustion Chamber: The basic geometry of combustor is derived by the need for its length and frontal area to remain within limits set by other engine components, the necessity of a diffuser to minimize pressure loss, and the requirement of a liner to provide stable operation over a wide range of air/fuel ratios. In spite of more arduous operating conditions like high pressure, temperature, and inlet velocity these days combustors continue to exhibit 100 percent combustion efficiency over the normal working range, demonstrate substantial reductions in pressure loss and pollutant emissions, and allow a liner life that is significantly longer than those of many other engine components. Despite these advances, the challenge to ingenuity in design is greater than even before. New concepts and technology are needed to satisfy current and projected pollutant emission regulations and to respond to the growing emphasis on engines that can utilize a much broader range of fuels. This change of emphasis has not been accompanied by relaxation of more conventional requirements of durability, pattern factor, and relighting capability. And sizing of combustor may now be determined by pollutant considerations. The desired performance requirements, in terms of higher engine/thrust ratio and lower specific fuel consumption, will call for higher turbine inlet temperatures and closer adherences to the design temperature profile at the turbine inlet. At the same time the demand for greater reliability, increased durability, and lower manufacturing, development and maintenance costs seems likely to assume added importance in the future. To meet these challenges, designers have searched for concepts that would simplify both the basic design data and methods of fabrication. This search has led to the development of advanced cooling configurations and the increased use of refractory coatings within the combustion system.

1.2 Types of Combustors

Tubular Chambers: A tubular chamber is comprised of cylindrical liner mounted concentrically inside a cylindrical casing. Most of the jet engines featured tubular chambers, usually in numbers varying from seven to sixteen per engine. However, for the majority aircraft applications, the tubular system is too long and heavy results in an engine of large frontal area and high drag. Annular Chambers: In this type an annular liner is mounted concentrically inside an annular casing, it is an ideal form of chamber, since its clean aerodynamic layout results in a compact unit of lower pressure loss than other chamber designs. The undesirable outcome of the annular systems is that a slight variation in the inlet velocity profile can produce a significant change in the temperature distribution of the outlet gases. And test bed development of annular chambers presents serious difficulties, owing to the very high cost of supplying air at the levels of pressure and temperature and in the amounts required to test large annular combustion chambers at full-load conditions.

Tuboannular Chambers: In the tuboannular chamber, a group of cylindrical liners is arranged inside a single annular casing. Compared with the annular design, the tuboannular chamber has an important advantage in that much useful chamber development can be carried out with very modest air supplies, using just a small segment of the total chamber containing one or more liners.

Liner of combustor: From analytical view point liner comprised of three zones. Primary, intermediate, and dilution zone.

Primary Zone: The function of primary zone is to anchor the flame and to provide sufficient time, temperature, and turbulence to achieve essentially complete combustion of the fuel.

Intermediate Zone: The main function of intermediate zone is to provide conditions that are conductive to recombination and thus to the elimination of dissociated products from gases entering the dilution zone. Intermediate zone also serves as an extension of the combustion zone under conditions for which combustion performance is limited by evaporation/reaction rates.

Dilution Zone: The role of the dilution zone is to admit the air remaining after the combustion and wall-cooling requirements have been met, and to provide an outlet stream with mean temperature and a temperature distribution that are acceptable to the turbine. The dilution air is introduced through one or more rows of holes in the liner walls.

1.3 Combustion: Combustion is one of the most important processes in engineering, which involves turbulent flow, heat transfer and other complicated physical and chemical processes. Combustion is a phenomenon through which the energy trapped in various fuels is converted from chemical form to heat (and light) form. The fuel used in industrial and domestic combustion equipment can occur in any of the three naturally occurring phases (solids, liquids and gases). This fuel has to react with oxygen, occurring in gaseous form. Therefore it is also needs to be converted to gaseous form before undergoing combustion reactions. At a molecular level, the two reactants can undergo a change in their electronic configuration to form or break bonds that result in a chemical reaction. They have to be mixed thoroughly to carry out efficient combustion. Therefore bringing the two reactants, viz. fuel and Oxygen, in the close proximity of each other at molecular level, forms a challenging part of significant in situations designing any combustion equipment. In general chemical reactions are of low pressure, for example, in the determination of ignition and stability at high altitudes. However, under many conditions interest is focused not so much on the limits of combustion as on the structure, heat release rates, combustion products, and radiation properties of high temperature flames. Most fuels used in combustion applications are a mixture of several chemical species. Each of these species reacts with oxygen releasing its respective heat of reaction. These reactions do not occur as a single-step process, but constitute several elementary steps involving many intermediate species. Knowledge of all such steps and intermediate species is essential in understanding the combustion behavior of fuels.

1.4 Computational Fluid Dynamics (CFD)

The equations of fluid mechanics which have been known for over a century are solvable only a limited number of flows. The known solutions are extremely useful in helping to understand fluid flow but rarely can they be used directly in engineering analysis or design. The engineer has traditionally been forced to use other approaches. Many flows require several dimensional parameters for their specifications and it may be impossible to set up an experiment which correctly scales the actual flow. After recognizing the power of computers become popular, interest in numerical techniques increased dramatically. Solution of the equations of fluid mechanics on computers has become so important that it now occupies the attention of a perhaps a third of all researchers in fluid mechanics and the proportion is still is increasing. This field is known as computational fluid dynamics (CFD) In computational fluid dynamics (CFD), flows and related phenomena can be described by partial differential equations which cannot be solved analytically except in special cases. To obtain an approximate solution numerically, we have to use a discretization method which approximates the differential equations by a system of algebraic equations, which can be solved on a computer. The approximations are applied to small domains in space and/or time so the numerical solution provides results at discrete locations in space and time. CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial applications

Present work discussed the effect of oxygen percentage in combustion air on combustion parameter and numerical simulation using Fluent. Oxygen percentage is varied from 10% to 30% in steps of 10%. For all simulation equivalence ratio is kept constant for all cases considered here.

II. MATHEMATICAL MODEL

In this work we used the following models for the numerical calculations: (a) turbulent flow, with turbulent model of RNG k-ε applied with a standard wall functions for near wall treatment; (b) for the chemical species transport and reacting flow, the eddy-dissipation model with the diffusion energy source option. The following assumptions are made: (a) the flow is steady, turbulent and compressible; (b) the mixture (propane-air) is assumed as an ideal gas; (c) no-slip condition is assumed at the burner element walls. The governing equations for mass, momentum and energy conservation, respectively, for the two-dimensional steady flow of an incompressible Newtonian fluid are:

Mass conservation equation:

$$\nabla.(\rho u_i Y_i) = -\nabla.J_i + S_i$$

with

$$J_i = -(\rho D_{i,m} + \mu_t / Sc_t)\nabla Y_i$$

and

$$D_{i,m} = \frac{1 - Y_i}{\sum Y_j / D_{i,j}}$$

Where ρ is the density, u_i is the fluid velocity, Yi is the local mass fraction, Ji is the diffusion flux, Si is the rate of creation by chemical reaction, Di,m is the diffusion coefficient, μt and Sct are the turbulent viscosity and Schmidt number, respectively.

Momentum conservation equation:

$$\nabla.(\rho u_i u_j) = -\nabla.P + \nabla.\tau_{eff}$$

with

$$\tau_{eff}| = \mu(\nabla u + \nabla u^T) - 2/3\nabla.u\delta$$

where τ_{eff} is the stress tensor, μ is the molecular viscosity and **δ** is the unit tensor.

Energy conservation equation:

$$\nabla.[u_i(\rho E + P)] = \nabla.\left(k_{eff}\nabla.T - \sum_j h_j J_j + u_j \tau_{eff}\right) + S_h$$

with

$$E = h - \frac{P}{\rho} + \frac{u_i^2}{2}$$

Where E is the energy, P is the pressure, k_{eff} is the effective conductivity, S_h is the source of energy and h is the sensible enthalpy. In this work, the combustion of propane with air is modelled with one-step reaction mechanism. The reaction mechanism takes place according to the constraints of chemistry and it is defined by

$$C_3H_8 + \frac{5}{\phi}O_2 + \frac{5}{\phi}\frac{100-\gamma}{\gamma}N_2 \rightarrow 3CO_2 + 4H_2O + \frac{5}{\phi}\frac{100-\gamma}{\gamma}N_2 + 5\frac{1-\phi}{\phi}O_2$$

where $\phi\,(= 5[M_{O2} + (100-\gamma)/\gamma\, M_{N2}\dot{m}_{fuel}]/(M_{fuel}\dot{m}_{air}))$

is the equivalence ratio and m_{fuel} and m_{air} are fuel and mass flow rates, respectively and γ is oxygen percentage in air.

$$R_{sto} = 5\frac{M_{O_2} + (100-\gamma)/\gamma\, M_{N_2}}{M_{C_3H_8}}$$

$$u_{air} = \frac{R_{sto}}{\phi}\frac{\rho_{fuel} A_{fuel}}{\rho_{air} A_{air}} u_{fuel}$$

		u_{air} (m/s)
γ (%)	R_{sto}	Φ=0.5
10	32.22	56.436
20	16.34	28.621
30	11.01	19.34

III. COMPUTATIONAL MODEL

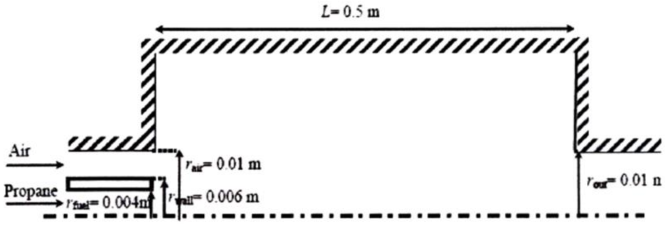

Fig. 1 Scheme of burner analyzed [1]

A two-dimensional burner element was designed using Gambit package as pre-processor. A turbulent model of RNG k-ε was applied with a standard wall function.

3.1 Geometry and Mesh Generation

Grid generation represents a major challenge for CFD analysis. It is a time-consuming task and, in spite of steady advances in automatic mesh generation, it still requires the skill of a CFD practitioner to yield a suitable mesh. The choice of the type of grid depends on geometrical complexity and on physics. The grid was smoothed using the swap/smooth options in both codes.

3.2 Gas Flow Simulation

For gas simulation a propane-air mixture was used with the following physical values: h_{amb} = 20 W/m K, Tin =T_{amb} =T_{ref}=300 K, p= 101325 Pa and air ρ = 1.225 kg/m^3 and ρC_3H_8 =1.91 kg/mρ at the air and fuel inlet, respectively. The thermal properties (Cp, μ and κ) of the propane and species are function of temperature. The propane density at the fuel inlet and the molecular weights, enthalpies and lower heating values of reactant and product species are taken from the material property database given by Fluent Inc. The ranges of the simulation values are: φ=0.5, and γ = 10%, 20%, 30%.

3.3 Fluent Modeling

The Fluent modeling is based on the two-dimensional conservation equations for mass and momentum. The differential equations are discretized by the Finite Volume Method and are solved by the SIMPLE algorithm. As a turbulence model, the k-ε was employed. The Fluent code uses a structured mesh, on which the conservation equations for mass, momentum and energy are discretized. The k-ε model describes the turbulent kinetic energy and its dissipation rate and thus compromises between resolution of turbulent quantities and computational time. No-slip condition is assumed at the burner walls.

4. Validation

Fig. 2 to Fig. 4 shows the comparison of the temperature distribution in the combustor. The counters reported by C.E.L. Pinhoa et al [1] and obtained in present numerical simulation are comparable. From fig. it is observed that as oxygen percentage increases, the maximum temperature increases inside the combustor.

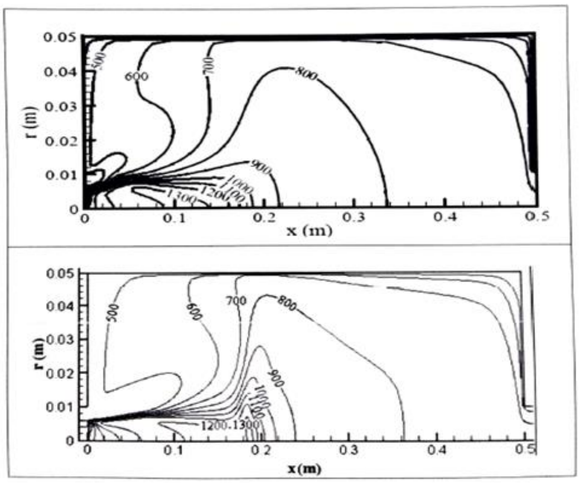

Fig. 2 Temperature profile (φ =0.5 and γ = 10%)

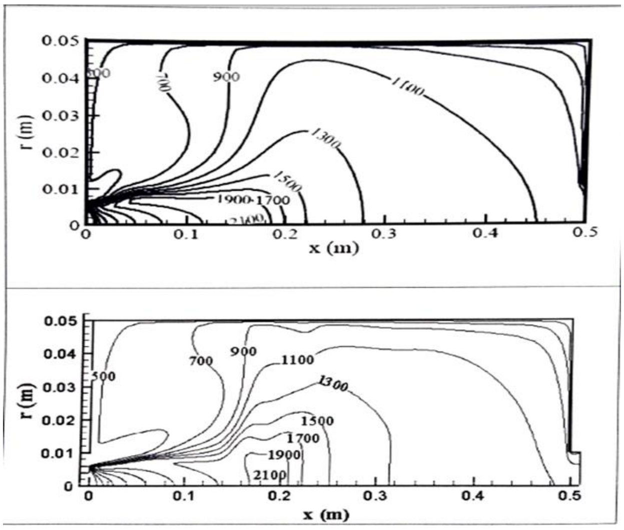

Fig. 3 Temperature profile (φ =0.5 and γ = 20%)

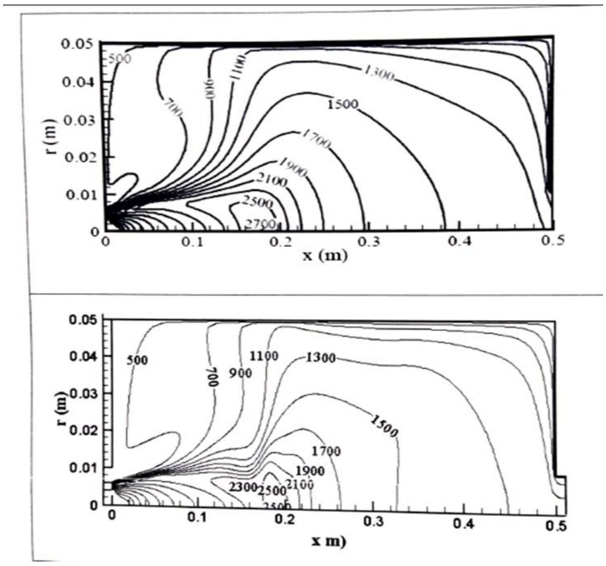

Fig. 4 Temperature profile (φ =0.5 and γ = 30%)

IV. RESULT & DISCUSSION

To study the effect of oxygen percentage in combusting air on combustion parameter numerical simulation is done using Fluent. For all simulation equivalence ratio is kept constant. Oxygen percentage is varied from 10% to 30% in steps of 10%. Variation of axial and radial velocity in radial direction along combustor length is plotted & discussed in following session. Temperature profile is plotted at different location along length of combustor. Mass fraction of fuel and oxygen is obtained at various stations to see the effect of O_2 % in combustor air. All these flow parameter are plotted at different Locations as shown in fig. 5.

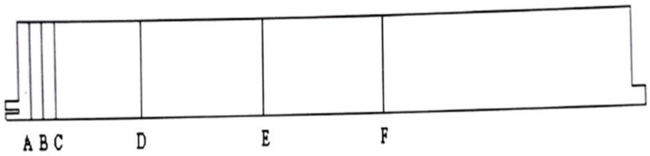

Fig. 5 Different locations in the combustor

4.1 Case 1: 10 % of O_2 in Combustion Air

4.1.1 Axial velocity profile

Axial velocity at different location A, B, C, D, E and F are plotted as shown in fig 6. Axial velocity is observed positive at all station except at location A and E. At location A axial velocity becomes negative near combustor axis. This is attributed to presence of strong recirculation region near fuel inlet. This recirculation zone is due to large value of combustion air velocity at inlet. This high value of air velocity is due to lesser O_2% in air. For constant equivalence ratio if O_2% is less than there is a need to increase mass flow rate. At location A and D peak value of axial velocity is obtained at the vicinity of combustor axis. This indicates presence of flame surface. At location E, maximum value occurs at the axis. This indicates that flame surface is converged at the combustor axis before location E.

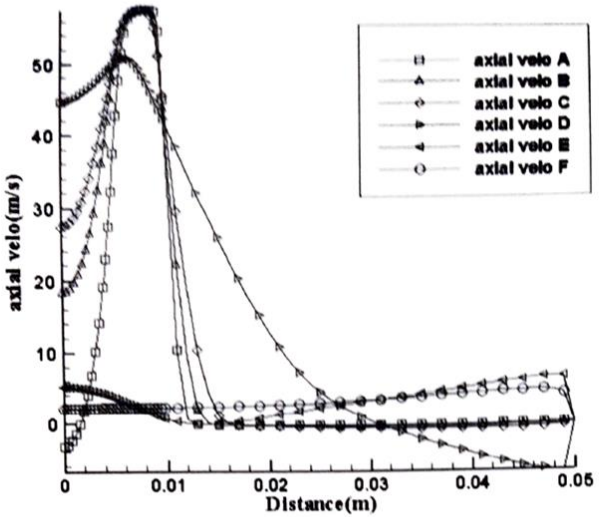

Fig.6 Axial velocity profile (10 % of oxygen)

4.1.2 Radial velocity profile

Radial velocity at different location A, B, C, D, E and F are plotted as shown in fig.7. For station A, the radial velocity is negative near the center because of the recirculation created due to high inlet velocity as discussed in axial velocity profile. For locations A, B, C, D, E the variations in the radial velocity are observed due to the recirculation created. For location E onwards, along the length of the combustor, the radial velocity becomes nearly zero which shows that the flame is converged and there is no inward and outward diffusion of O_2 and fuel. For all the locations radial velocity becomes zero at the wall of the combustor.

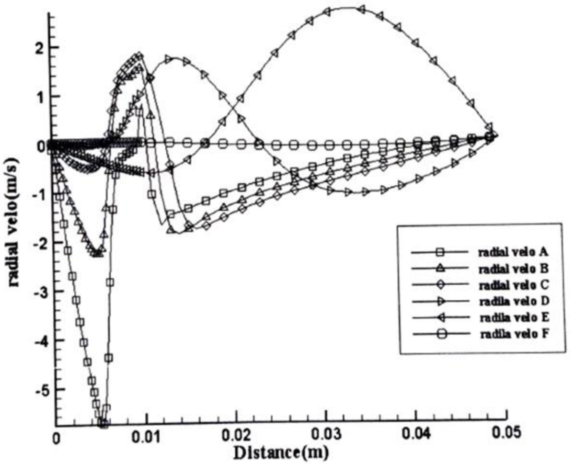

Fig.7 Radial velocity profile (10 % of oxygen)

4.2 Case 2: 20 % of O_2 in Combustion Air

4.2.1 Variation of velocity profile

Profile of velocity distribution at all station shown in fig. 8 is comparable with that of case- l. Axial velocity is positive at all station. At D negative axial velocity is observed at wall due to recirculation. Peak Values of axial velocity at different locations are almost half of that of case-1. This is due to increase O_2 % which reduces inlet air velocity for same equivalence ratio. Axial velocity at location A becomes positive as the recirculation zone created in previous case is not present at the axis in this case. This is due to the decreased air velocity. Radial velocity profiles shown in fig.9 are comparable for the locations A, B, and C with that of case- I. Negative value of radial velocity at location A is reduced by half because of the weak recirculation zone. Positive peaks at A, B, C are comparable with that of case- I.

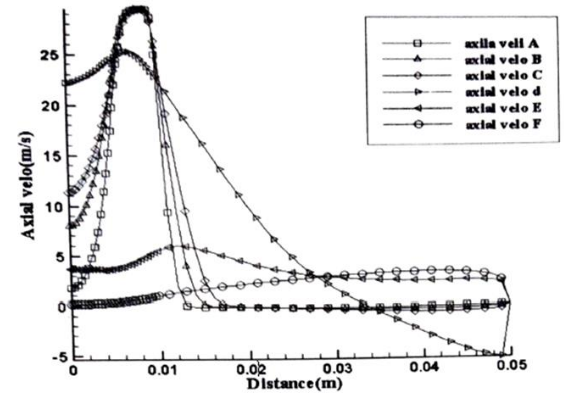

Fig.8 Axial velocity profile (20 % of oxygen)

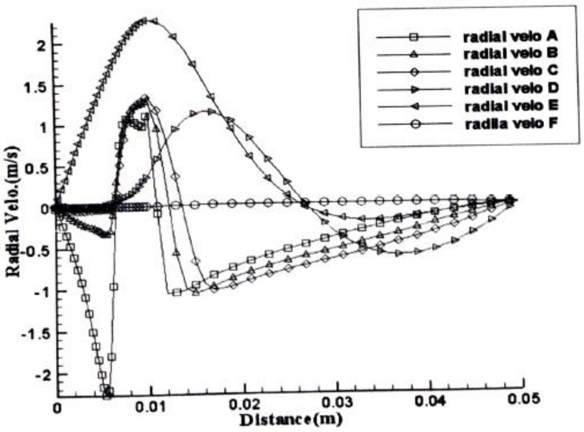

Fig.9 Radial velocity profile (20 % of oxygen)

4.3 Case 3: 30 o/e of O_2 in Combustion Air

4.3.1 Variation of velocity profile:

Nature of axial velocity profile shown in fig.10. Trends are comparable for all location with that of case-2. Peaks are reduces due to increased O_2 % in air. Axial velocity at location A becomes positive as the recirculation zone created in case I is eliminated in this case. The axial velocity at location E is higher than that of the case 2. This shows that the flame has penetrated further in axial direction compared to case-2. Profile of radial velocity is shown in fig.11. For case-2 and case-3 of radial velocity profile is comparable for the locations A, B, and C. Negative value of radial velocity at location A is reduced further compare to case-2 as effect of recirculation zone is negligible. Negative peaks for locations B and C are further reduced. Positive peak values of radial velocity for locations A, B and C are comparable. Peak value for location D and E are away from the axis which shows that the flame surface further diffused in radially outward direction.

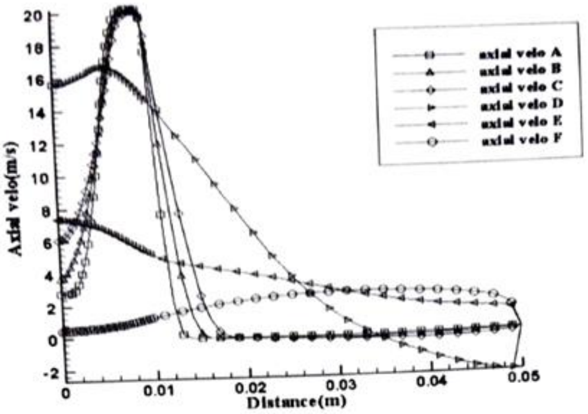

Fig.10 Axial velocity profile (30 % of oxygen)

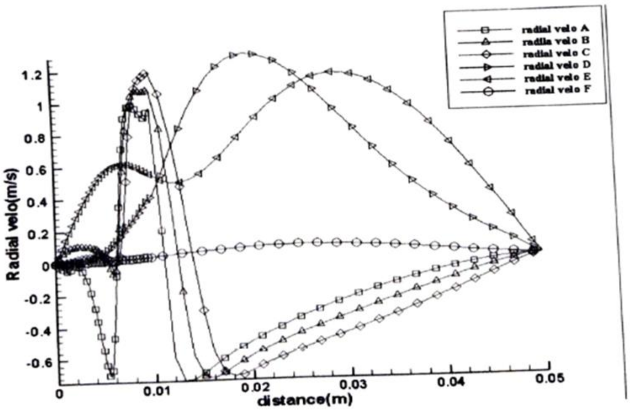

Fig.11 Radial velocity profile (30 % of oxy gen)

V. CONCLUSION

An attempt has been made to investigate the effect of oxygen percentage on axial and radial velocity inside coaxial- combustor. Numerical simulation is done using Fluent. Oxygen percentage is varied for constant equivalence ratio from 10% to 30% in steps of 10%. Combustion get enhanced with increased percentage of oxygen. Strong recirculation zone is observed at fuel inlet in case of 10 % oxygen. In this case very high velocity creates the recirculation zone at the entry of fuel in to the combustor. This recirculation zone is responsible for low concentration of fuel along combustor axis.

VI. REFERENCES

[1]. C.E.L. Pinho, J.M.P.Q. Delgado R. Pilao, J. Conde and C. Penho2, "Numerical Study of Propane-Air Mixture Combustion in a Burner Element", Defect and Diffusion Forum Vols. 273-276 (2008) p.144-149.

[2]. G. Wecel and R.A. Bialecki: BEM Solution of the Radiative Heat Transfer within Natural Gas Fired Combustion Chamber", Combust. Sci. and Tech. Vol. 178 (2006), p. 1413

[3]. Mustafa Ilbas, "The effect of thermal radiation and radiation models on hydrogen-hydrocarbon combustion Modeling" International Journal of Hydrogen Energy Vol.30 (2005) p.1113 - 1126

[4]. Fluent Incorporated: FLUENT User's guide version 6.2 (2005).

[5]. V. Yakhot and S.A. Orszag: Renormalization Group Analysis of Trbulance", J. Sci. Comput. vol.1 (1986), p.151

[6]. Mustafa ,Ilbas,, Ilker Yilmaz, Yuksel Kaplan, "Investigations of hydrogen and hydrogen-hydrocarbon composite fuel combustion and NOx emission characteristics in a model combustor" International Journal of Hydrogen Energy 30 (2005), P· 1139 - 1147

[7]. H.K. Versteeg and w. Malasekera: "An introduction Computational Fluid Dynamics" (Published by Pearson Education, Delhi, 2008)

[8]. D.P. Mishra: "Fundamental of combustion" (Prentice-Hall of India Pvt. Ltd., New Delhi, 2008)

Cite this article as :

Prakashkumar R. Patel, Ambaliya Sanjaykumar D, " Numerical Investigation on Variation of Axial and Radial Velocity Profile Inside CAN Type Combustion Chamber , International Journal of Scientific Research in Science, Engineering and Technology(IJSRSET), Print ISSN : 2395-1990, Online ISSN : 2394-4099, Volume 3, Issue 5, pp.831-838, July-August-2017.
doi : https://doi.org/10.32628/IJSRSET1735145