IOP Conf. Series: Materials Science and Engineering **377** (2018) 012151 doi:10.1088/1757-899X/377/1/012151

A Numerical Study on variation of Flow Pattern of Fluid Passing through three Different Modified Dump Combustors

Chandan Mandal* **and Somnath Chakrabarti**

Department of Mechanical Engineering, Indian Institute of Engineering Science and Technology, Shibpur, Howrah-711103, West Bengal, India.

*Corresponding authors e-mail: mandalc39@gmail.com

Abstract. In this paper, a numerical study on flow characteristics of fluid passing through a sudden expansion plain dump combustor and modified dump combustors having a central restriction of three different shapes at a certain distance ahead the throat has been carried out. The two dimensional, steady differential equations for conservation of mass, momentum and the governing realizable k-ϵ equations are solved for the Reynolds number 1.2×10^5, aspect ratio (AR) of 3 and 100% central restriction. The variations of streamline contours, re-circulating zones and axial velocities along the radial distance at various axial positions have been studied. After overall study, it is observed that the adverse pressure gradient between central and wall region at the outlet casing is more. Re-circulation zone (RZ) formation in modified dump combustors is more compared to a plain sudden expansion combustor. The fluctuation of radial profile of axial velocity is more at any location for the fillet shape restriction compared to other two shapes of restrictions (rectangular and hemispherical). The modified combustors may perform in efficient manner in terms of complete combustion and less pollutant emission due to formation of more re-circulating bubbles throughout the outlet casing of the combustor.

1. Introduction

In combustion system, modeling of a combustion chamber is very essential for stable complete combustion and to avoid a large total pressure loss. For an efficient combustion with less number of pollutants, flow pattern plays an important role in controlling the combustion process by influencing the fuel-air mixing. Re-circulating bubbles have significant role on determining the efficiency of the dump combustor. The said bubbles are helpful for uniform mixing, flame stabilization and also reducing the NO_x emission. Sudden expansion configurations having some modifications in its geometry have inherence effects on flow pattern, wall pressure and axial velocity. The placement of central restriction with different shape at the exit zone (i.e., zone between throat section and outlet section), may be considered as an important aspect for creation of more re-circulating bubbles in that zone. Central restriction also helps to fluctuate the turbulent kinetic energy and breaks the flow vortex to form a central re-circulating zone (CRZ) into the combustor. This modified combustor is required in many practical applications, like chemical process industry, mixing chamber, gas turbine, combustion chamber, diffuser etc.

To understand the fluid dynamics phenomena by incorporating different type of restriction or modification in geometry Mandal *et al*. [1] have numerically analysed the performances of a sudden expansion diffuser by placing a fence near the throat region. They have compared their model performances with a simple sudden expansion configuration without fence by changing the fence angle and its position for low Reynolds numbers. They have reported that at high Reynolds number, fence is more beneficial to create re-circulation zone. Gobbato *et al*. [2] have experimentally carried out the effect of the Reynolds number and the basic design parameters on the isothermal flow field of low-swirl combustor. By using a hot-wire probe they have performed the tests at a high Reynolds number for a combustor of high expansion ratio. They have incorporated a swirler at the inlet casing of the combustor and measured the variation of tangential and axial velocity at various axial positions and compared the results with open literature results. They also reported that for less values of

expansion ratio (ER) the outer re-circulating zone (ORZ) moves closer to the jet with a higher velocity of the fluid particle within the ORZ. Mondal *et al.* [3] have numerically analysed influence of side wall expansion angle on flow pattern in a model combustor calculated with *k–ε* model. Numerically they have investigated variation of axial velocity with radial distance at various axial positions for different side wall expansion angle. They have identified the CRZ and shown the variation of streamline pattern in the combustor and presented the detailed characterization of streamline in terms of size, location and strength. They have also shown the radial distribution of turbulent kinetic energy, variation of re-circulated mass within the central recirculation zone with different inlet swirl numbers and different side wall expansion angles. Ghose *et al.* [4] have numerically studied the effect of dome shape on static pressure recovery in a dump diffuser at different inlet swirl numbers by using realizable k-ε model. They have considered three different dome shapes (hemispherical, ellipsoidal and vertical ellipsoidal) for their simulation and observed that some re-circulating bubbles are formed at the corner region of dump diffuser, whose size and intensity are dependent on different dome shapes and swirl numbers. Raj and Ganesan [5] have performed both numerically and experimentally the effect of various parameters on flow development for different swirl flow. An appropriate standard turbulence *k–ε* model has been selected for their prediction. Geometry and mesh have been done in GAMBIT and mass and momentum equations have been solved in CFD by using SIMPLE algorithm. They have shown the variation of axial velocities with radial distance for different vane angle and reported size and characteristic of the re-circulation zone. Das and Chakrabarti [6] have carried out a 2-D numerical study on flow characteristics for four different type annular dump combustor models. They have considered a fixed Reynolds number 200, AR=2, CR=40% and studied the variation of streamline contour, average static pressure and stagnation pressure distribution. They have observed that sudden expansion with central restriction and fence performs better in terms of average static pressure distribution along the axis and average stagnation pressure drop variation become more effective for corner re-circulating bubbles. Walker *et al.* [7] have done their experiment on a hybrid diffuser for gas turbine combustor and investigated the governing flow mechanisms. They have also measured the axial velocity at outlet guide vane (OGV) exit and predicted that a local stream wise momentum transfers from the accelerating bleed flow to the diffused mainstream flow. They have reported that stream wise momentum transfer is responsible to create a reverse negative velocity.
Since, there are no much numerical or experimental work on the performance and behaviour of fluid flow of a modified dump combustor, therefore, in this paper an attempt has been made to investigate the effect of central restriction shape on flow pattern and axial velocity at different axial positions. After comparison of the results between the flow characteristic of a plain and a modified dump combustor under turbulent flow condition for a Reynolds number $1.2\text{x}10^5$, it may be said that modified dump combustors are more efficient than a plain dump combustor.

2. Mathematical Formulation

2.1. Computational Domain
A schematic diagram of the computational domains for the flow through a plain dump combustor and a modified dump combustor configuration with various restriction shapes are shown in figures 1 (a), (b),(c) and (d). The size and shape of the combustor considered in the present work are taken from Ko and Sung [8]. The length of the Inlet pipe (L_i) and the casing (L_{ex}) are 0.08 m and 0.3 m respectively. The diameter of inlet duct (D_i) and outlet duct (D_o) are 0.04 m and 0.12 m respectively. For figure 1 (b) and (c) the length (L_r) and diameter (D_r) of central restriction have been chosen as 0.06 m and 0.04 m respectively. The fillet angle (θ) is taken as 15^o. For figure 1 (d) the radius (R_r) of the hemispherical dome shape is taken as 0.02 m from Ghose *et al.* [4].

2.2. Governing Equations for Turbulent Flow

The flow is considered to be steady, two dimensional, axis-symmetric and turbulent. The fluid is considered to be Newtonian and incompressible. Under these assumptions, the governing equations become,

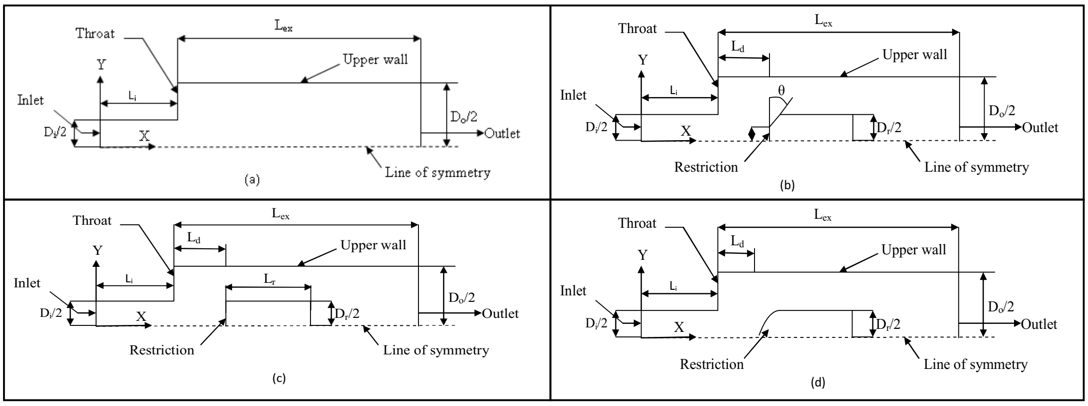

Figure 1. Schematic diagram of the computational domain (a) plain dump combustor, (b) and (c) and (d) modified dump combustor configuration with restriction of fillet, rectangular and hemispherical dome shape respectively.

Continuity equation: $\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$ (1)

Momentum equation in X-axis: $\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_x}{\partial x}$ (2)

Where, τ_x is the turbulent stress tensor along x direction and $\tau_x=\left[\mu\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}\right)\right]-\frac{2}{3}\mu\frac{\partial u}{\partial x}$

Momentum equation in Y-axis: $\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_y}{\partial y}$ (3)

Where, τ_y is the turbulent stress tensor along x direction and $\tau_y=\left[\mu\left(\frac{\partial v}{\partial x}+\frac{\partial v}{\partial y}\right)\right]-\frac{2}{3}\mu\frac{\partial v}{\partial y}$

The governing realizable k-ϵ equations representing the turbulent properties are as follows,
k- Equation:

$$\rho\left(u\frac{\partial k}{\partial x}+v\frac{\partial k}{\partial y}\right)=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x}+\left(\mu+\frac{\mu_t}{\sigma_k}\right)-\frac{\partial k}{\partial y}\right]+\rho G-\rho\varepsilon$$

ε- Equation:

$$\rho\left(u\frac{\partial \varepsilon}{\partial x}+v\frac{\partial \varepsilon}{\partial y}\right)=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}+\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)-\frac{\partial \varepsilon}{\partial y}\right]+\rho C_1 S\varepsilon+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G-\rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}}$$

Where, G is the production term and this term represents the transfer of kinetic energy from mean flow to turbulent motion through the interaction between the turbulent fluctuations and the mean flow velocity gradients. ε is the rate of dissipation of turbulent kinetic energy due to viscous effects.The turbulent viscosity, $\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}$

Here, $C_{1\varepsilon}$, C_2, C_μ, σ_k and σ_ε are the empirical turbulent constant. The values are considered according to the Launder and Spalding [9]. The values of $C_{1\varepsilon}$, C_2, C_μ, σ_k and σ_ε are 1.44, 1.9, 0.09, 1.0 and 1.0 respectively.

2.3. Boundary Conditions

Four different types of boundary conditions have been considered for this present work. These are as follows,

- At the walls: No slip condition, i.e. velocity component at the wall is zero.
- At the inlet: Axial velocity is specified and transverse velocity is set to zero, i.e., u=specified and v=0.
- At the exit: Constant pressure boundary has been taken.
- At the line of symmetry: the normal gradient of the axial velocity and the transverse velocity are set to zero.

2.4. Numerical Procedure

The geometry and mesh of the computational domain have been made in commercial CFD software FLUENT 16.0. The partial differential equations (1) – (3) are discretised by a control volume based finite difference method. Power law scheme is used to discretise the convective terms, Patankar [10]. The continuity and momentum equations are solved by using SIMPLE algorithm along with the upwind scheme. Uniform and staggered grid arrangements have been selected for present numerical analysis. In this study, the considered working fluid is air. The density of air, ρ is taken as 1.225 kg/m^3. The viscosity of air, μ is taken as 1.7894×10^{-5} kg/m-s. Inlet velocity is obtained from the considered value of the Reynolds number by using the relation, $Re=\rho u_i D_i/\mu$. The dynamic pressure at inlet is obtained by using the relation, $P_{d\text{-}in}=\rho u_i^2/2$, where u_i is the velocity at inlet.

3. RESULTS AND DISCUSSION

The present numerical studies have been carried out to investigate the effect of different central restriction shape on the flow characteristic of fluid passing through modified dump combustors and compared with a simple plain sudden expansion combustor. Variation of flow pattern, re-circulating zone formation and axial velocity along the radial distance at different location for a turbulent flow are mainly discussed.

3.1. Variation of flow patterns for different restriction shape

The flow pattern of streamline is shown in figure 2 that reveals formation of a corner recirculation zone and a central recirculation zone immediately downstream of the inlet. However some recirculation zone are also formed near the restriction wall and just after the second expansion at the outlet casing of modified combustors with central restriction. The central recirculation bubbles grow in larger size for a plain sudden expansion configuration shown in figure 1(a). Whenever a rectangular or a fillet edge restriction is incorporated at the central region shown in figure 1(b) and (c), then two more recirculation zone are formed near the restriction wall due to adverse pressure gradient at the downstream. Further changing the restriction as a hemispherical dome shape shown in figure 1(d), it is noticed that due to high reverse velocity, a combined longer recirculation zone is formed near the throat and the second contraction region.

3.2. Variation of axial velocity with radial distance

The variation of the mean axial velocity at different axial positions (i.e., X/D_i of 2.5, 3.5, 4.5 and 5.5) are shown in figures 3(a), (b), (c) and (d) for all the configurations. Figure 3(a) shows that for all the configurations, the axial velocity remains constant up to a certain radial distance ($r/R_i=1$) after that it suddenly decreases to zero or negative in magnitude due to difference of flow kinetic energy. Due to this, the corner re-circulating bubbles are formed near the throat region. Radial profile of axial velocity

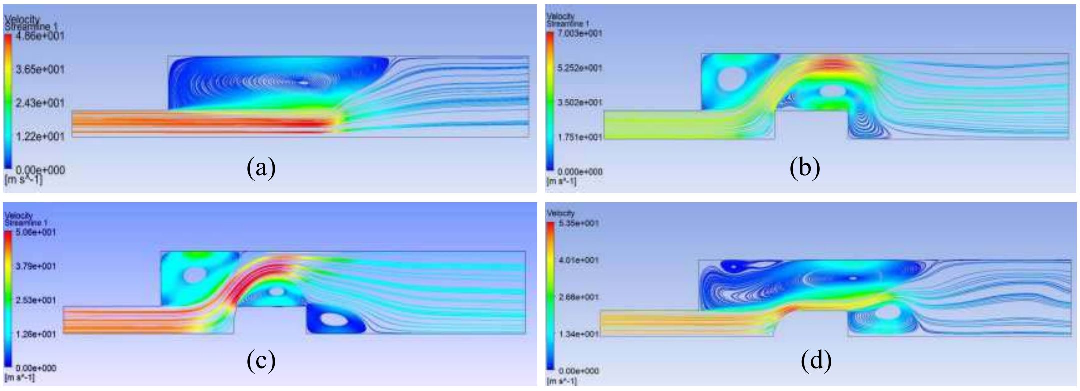

Figure 2. Variation of streamline contours of plain combustor (a) and modified dump combustor with restriction of rectangular (b), fillet (c) and hemispherical dome (d) shape respectively.

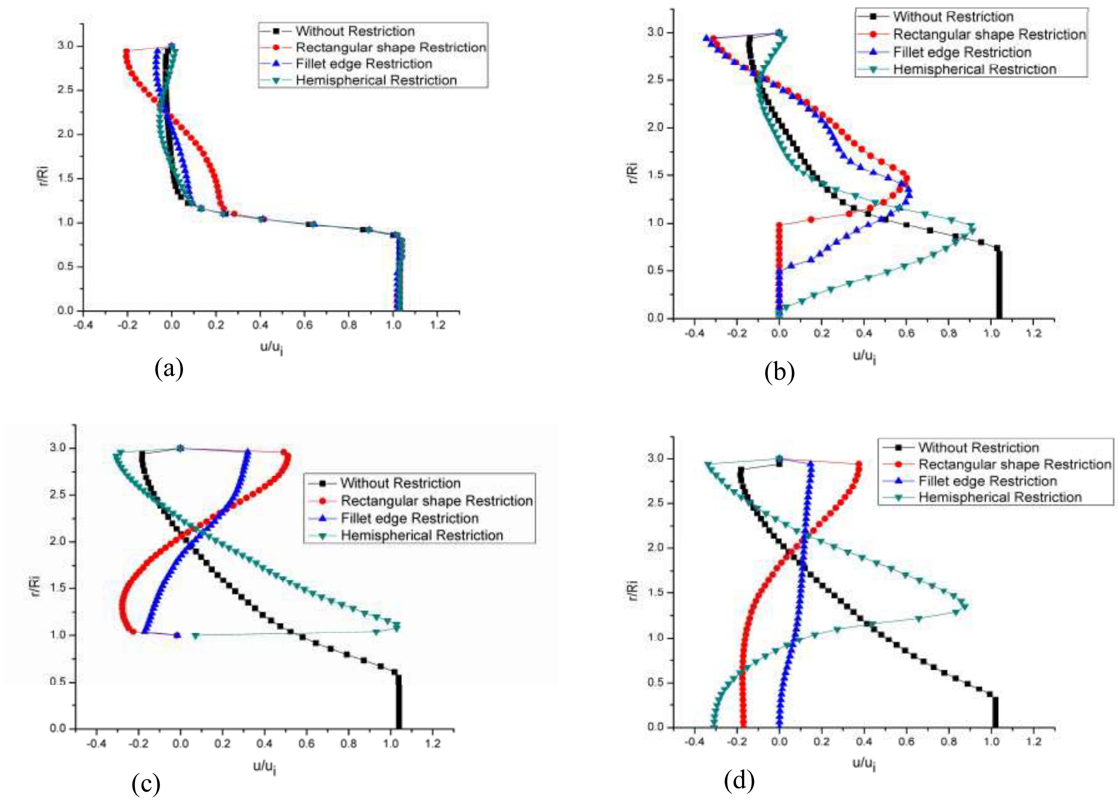

Figure 3. Variation of axial velocity along the radial distance at X/D_i=2.5 (a), X/D_i=3.5 (b), X/D_i=4.5 (c) and X/D_i=5.5 (d) respectively.

just before the restriction and at the starting of second contraction in figure 3(b), shows that near the inlet wall of restriction velocity is zero after that it goes to maximum value and then goes to minimum value near the wall region. In case of rectangular and fillet shape restrictions, axial velocities are negative near the upper wall of the outlet casing and form a wide RZ that spread form the throat section to the entry of second contraction section in the corner region.

From figure 3(c), it is noted that over the restriction wall, axial velocities are negative for rectangular and fillet shaped combustor model. These velocities with increase in the radial distance and further decrease near the wall due to viscous resistance. The said velocities create a RZ over the restriction wall. For hemispherical shaped restriction, velocity is sharply decreased from upper wall of restriction to the casing wall. At outlet casing just after the second expansion, it is observed from figure 3(d), that the reverse velocity near the wall of hemispherical restriction increases to maximum and then it goes to negative in magnitude due to sudden expansion at the outlet casing. This reverse velocity forms two RZs. Velocity profile in case of without restriction dump combustor remains approximately same in all locations. Fluctuation of axial velocity with the radial distance is more at any axial positions for both rectangular and fillet shape restrictions. For said two cases, formation of RZ is also more. These additional re-circulating bubbles at RZ may offer uniform mixing and complete combustion into the combustor.

4. Conclusions

A numerical investigation is performed to study the flow characteristics and performance analyses of a simple plain combustor and a modified dump combustors with different shape of restrictions for a turbulent flow condition. The observations are summarized below,

- The performance of modified dump combustors with central restriction is always more than that of a simple sudden expansion plain combustor under turbulent flow condition.
- From the characteristic of the flow pattern of streamline, it is evident that more re-circulating bubbles form into the complete RZ in case of fillet edge restriction compared to other three configurations.
- Fluctuation of kinetic energy with the radial distance at different positions, create an adverse pressure gradient which is responsible to form RZ.

References

[1] Mandal D K Bandyopadhyay S Chakrabarti S 2011 A numerical study on the flow through a plane symmetric sudden expansion with a fence viewed as a diffuser *Int. J. of Engg. Sci. and Tech.* **3** 210-233

[2] Gobbato P, Masi M, Cappelletti A and Antonello M 2017 Effect of the Reynolds number and the basic design parameters on the isothermal flow field of low-swirl combustors *J. of Exp. Thermal and Fluid Sci.* **84** 242-250

[3] Mondal S, Datta A and Sarkar A 2004 Influence of side wall expansion angle and swirl generator on flow pattern in a model combustor calculated with *k–ε* model *Int. J. of Thermal Sci.* **43** 901–914

[4] Ghose P, Datta A and Mukhopadhyay A 2013 Effect of dome shape on static pressure recovery in a dump diffuser at different inlet swirl *Int. J. of Emerging Tech. and Advanced Engg.* **03** 465-471

[5] Raj R T K and Ganesan V 2008 Study on the effect of various parameters on flow development behind vane swirlers *Int. J. of Thermal Sci.* **47** 1204-1225

[6] Das T and Chakrabarti S 2016 A 2-D numerical study on flow characteristics for four different types annular dump combustor models *World J. of Modelling and Simulation* **03** 208-218

[7] Walker A D Denman P A McGuirk J J 2004 Experimental and Computational Study of Hybrid Diffusers for Gas Turbine Combustors *J. of Engg. for Gas Turbine and Power* **126** 717-725

[8] Ko S C and Sung H J 2002 Large-scale turbulent vertical structures inside a sudden expansion cylinder chamber *J. of Flow Turbulence and Combustion* **68** 269-287

[9] Launder B E and Spalding D B 1997 The Numerical Computation of turbulent flows *Computer method in Appl. Mech. and Engg.* **03** 269-289

[10] Patankar S V 1980 *Numerical Heat Transfer and Fluid Flow* (Washington DC: Hemisphere) p 126