CFD ANALYSIS OF TURBULENT FLOW IN AN AXI SYMMETRIC DUCT WITH SIDE INJECTION

G. Shantha [1], S. Aruna [2]

Department of Mathematics & Humanities, Mahatma Gandhi Institute of Technology Hyderabad, India.

Corresponding Author: shantha_maths@yahoo.com

ABSTRACT

A numerical reenactment has been done to consider the impacts of twin slanted side mass infusion with cross move through a roundabout pipe using modified $\boldsymbol{k}-\varepsilon$ model, considering smooth out ebb and flow impacts by adjusting the model constants. 1/seventh violent speed profile has been taken at the gulf. The effects of side weight injections have been analyzed in detail for the flow pattern of the main bulk liquid and two interconnected turbulent flows. Variating the primary and secondary injection angles visualize recirculatory flow formation. Axial velocity profiles at different locations and the variation in centerline speed along the duct have been examined with the variations in the injection angle. The effect on the recirculation size is more than the secondary injecting angle difference of the main injection angle.

1. INTRODUCTION

The side mass injection flow allowed normally or at an angle into a main turbulent bulk flow occurs in various industrial processes. To get better performance in a Gas Turbine combustor, mixing of air entering through the liner holes with high temperature combustion gasses is very much required to provide a more uniform temperature pattern at the outlet of the combustor to be compatible to the Turbine blades. The degree and rate of the mixing process is important in combustion applications because the burning efficiency and the exhaust composition depend on mass transfer. Several researchers have done theoretical, numerical and experimental works in this field studying parameters that include the penetration and mixing characteristics of cross turbulent flows. In most of the works the main duct fluid flow has been considered along with the side mass injection. Two cases are considered in the present inquiries. Also the side mass injection is present in the first case of the main flow. Where a partial main flow with side mass injection was considered as in the second case.

The picture. 1 shows the basic characteristics of the turbulent axi-symmetric flow through an inlet-closed, circular duct which was considered in the duct entrances from the porous wall from radial directions. There is no principle move through the channel and as the bay of the conduit is closed so it has been considered as a wall and the wall boundary conditions (no slip) has been applied at the inlet in the present analysis. The cylindrical coordinate $r-\theta-x$ has been considered. However the azimuthal component is absent owing to the fact that the geometry as well as the boundary conditions is axi-symmetric. The side injection velocity has been denoted by*Vinj* .

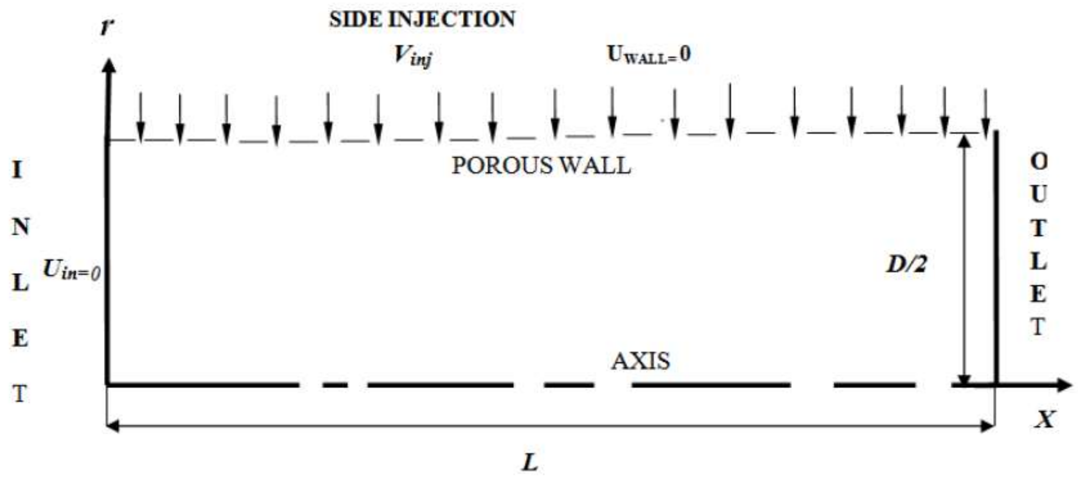

Figure 1, Schematic diagram for the circular axi-symmetric conduction by porous wall and closed inlet with lateral injection..

Fig. 2 shows the essential features of the axi-symmetric turbulent fluid flow through a partially opened enclosure, which has been considered with entries in the duct from radial directions through the porous wall. In the present analysis the inlet opening radius has been varied to investigate the flow pattern in the axi-symmetric circular duct.

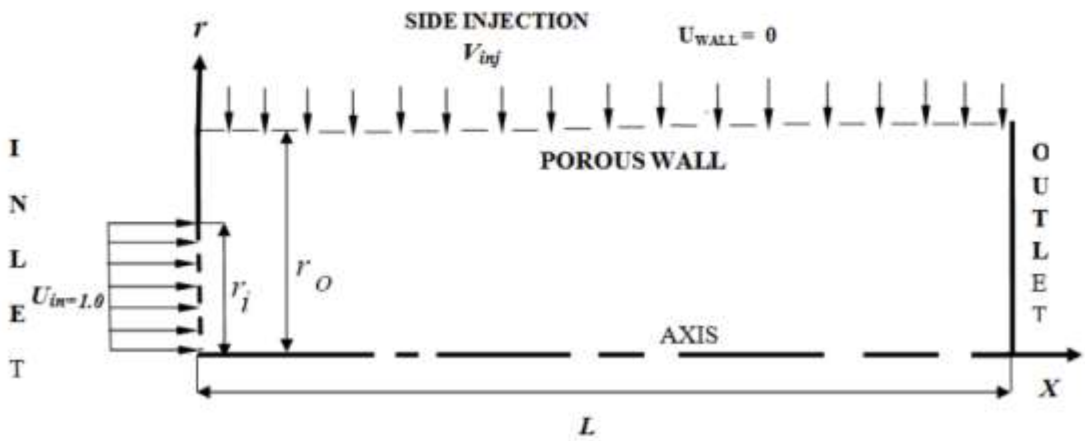

Figure 2, Schematic diagram of the axially symmetrical circular conduit with porous and partially opened inlet for side injections

2. LITERATURE REVIEW

The phenomena of side injection flow injected normally or at an angle into a main turbulent bulk flow occur in various industrial processes. These include for example the effluent operations where streams are mixed for the dilution and reduction of pollutant formations. For design consideration in the dilution zone of a gas turbine combustor, mixing of air entering through the liner holes with high temperature combustion gasses leaving the primary zone is very much required to provide a more uniform temperature pattern and rapid quenching for chemical reaction. Generally the objective is to rapidly obtain a homogeneous mixture of the injectant and mainstream. The degree and rate of the mixing process is especially important in combustion applications since burning efficiency and exhaust composition directly depend on mass transfer and reaction kinetics. Several experimental, numerical and theoretical works have been done by many researchers where the parameters studied include the side injection angle, the penetration and mixing characteristics of cross turbulent flows. A CFD study was performed to analyze the mixing potential of opposed rows of jets injected into a confined cross flow in a rectangular duct by Bain and Smith (1992). The mixing phenomena have been investigated considering straight, perpendicular slanted and parallel slanted slots. Yang and Shyu (1998) analyzed the numerical figures for the fluid flow and heat transfer properties of several pending slot jets with a tilted containment surface. The numerical result showed that the maximum pressure on the impinging surface move downstream while the inclination angle increased. Turbulent flow fields resulting from an oblique jet injecting from a rectangular side inlet duct into a rectangular main duct with an aspect ratio 3.75 without a forced crossflow were presented by Tong-Miin Liou *et al.* (1999). The main objectives were to study the effects of the side-jet angle (θ) and side-jet flow rate (Qs). The results showed that with increasing injection angle the size of the recirculation zone increases.

Woodfield *et al.* (2000) presented a study on numerical computation on recirculation flow structures in coaxial confined laminar jets. Their work showed that the formation of the recirculation zone is greatly influenced by Reynolds number and 'Craya-Curtet (Ct)' number under laminar flow conditions. The Craya-Curtet number is a significant boundary for portraying the re-course zones that are regularly connected with blending of planes. The blending procedure of violent streams inside a pipeline because of single and different transverse planes was broke down by Giorges et al. (2001). The numerical outcomes proposed that various planes guarantee better blending notwithstanding diminishing force necessities with an expanding number of planes. Synthetic blending in an open channel with numerous roundabout fierce planes were examined by Moawad et al. (2001). The trials were completed by shifting the proportion of speed of planes to that of cross stream, number of side infusion ports and the separation between the infusion ports. It was reasoned that by utilizing different planes, one can accomplish better blending in a shorter good ways from the fly infusion point than utilizing a solitary stream.

A line of planes releasing regularly into a restricted tube shaped cross stream was numerically examined utilizing the control-volume-based limited distinction strategy by Tao et al. (2002). Boundary varieties contemplated incorporate spout width, number of spouts, pipe sweep, fly and standard volume-stream, temperature proportion, and dynamic weight proportion. The qualities for infiltration profundity and spout dividing were portrayed for ideal blending. The two dimensional impinging roundabout twin stream with no cross stream was concentrated numerically and tentatively by Abdel-Fattah (2007). He presumed that the sub air district happens at the impinging plate and it increments emphatically by expanding Reynolds number and diminishes as the fly edge diminishes. To examine the blending and outflows in the RQL combustor plan to limit the development of NOX in gas turbine combustor Holderman and Chang (2008) researched the blending of various planes with a restricted subsonic cross stream. The trial results demonstrated that the quantity of openings significantly affected blending and the NOX outflow.

The conduct of single and various level gas stream infusions into a rectangular foaming fluidized bed were concentrated by Li et al. (2009) with three dimensional numerical reproductions. It has been demonstrated that the optional gas infusion firmly influenced the upper part over the infusion level. The impedance impacts of side planes with supersonic cross stream for a horizontal fly controlled rocket were reproduced numerically by Ashwin and Chakraborty (2010). From nitty gritty stream field examination it was seen that the age of pitching second occurred because of low weight area behind the fly spout and the typical power and pitching second apparently varied straightly with the fly weight proportion.

3. COMPUTATIONAL METHODOLOGY

A two-dimensional, turbulent and fully formed stream of diffuser inlet. The inlet speed of the air remains 14.61 m / s, while the Reynolds number is 5 to 104 based on the diffuser inlet. Originally, a commercial geometry program of cum grid generation is used to formulate three-dimensional spaces – 'Gambit" for axisymmetric diffusers with different half angles, with a constant area ratio of 1,6 for all cases. Table 1 displays half angles and corresponding mesh sizes.

Table 1: Axisymmetric diffuser calculation mesh dimensions.

Half angle (α°)	No. of Cells
6	135,000
10	112,500
18	110,600
90	99,500

The first grid point at 0.1 mm for the wall effect and separation is used to construct a boundary layer. The hexahedral meshing is done in such a way that the core region is ordered. The speed-entry limit condition and the pressure-outlet limit were established respectively at the inlet and outlet of the diffuser. A renewable group (RNG) k-00 is used for code validation. With any calculation of CFD flow the correct choice of turbulence models requires attention. This turbulence model is part of the model class of two equations in which transport equations are resolved for two turbulence amounts k and ε. Possibly the most basic and implemented maximum turbulence model is the k− ε model. It belongs to a variety of engineering questions, which are part of most commercial CFD codes (Pope, 2009). The concepts as well as the details have evolved in the course of time, as is the case in all turbulence models. But Jones and Launder (1972) have developed the 'standard' k− ε model appropriately, and Launder and Sharma (1974) offer enhanced values for model constants.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Two further equations, namely the transport equations for k and ε, are needed to close the model:

k-equation:

$$\frac{(\rho U_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\cdot\frac{\partial k}{\partial x_i}\right] = P_k - \rho\varepsilon$$

ε –equation:

$$\frac{\partial(\rho U_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\cdot\frac{\partial \varepsilon}{\partial x_i}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}P_k - C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

To accomplish assembly, a steady state-based verifiable solver is utilized. So as to accomplish more noteworthy accuracy in tests, the second-request upwind strategy was utilized to diminish the entirety of the conditions. A weight speed connection utilizing a SIMPLE calculation has been built up by speed-pressure coupling (Patankar, 1980). Low-unwinding factors were utilized for all Scarborough conditions (User Guide Fluent, 2006). Continuous control of residuals was carried out on continuity, x-speed, y-speed, z-speed, k and ε.

CONCLUSIONS

The extending of the axisymmetric diffuser cross-sectional area over different half corners illustrates how the bulk flow is transited into pressure-induced separation and the subsequent downstream flow re-alimentation. Various injeService Delivery Managerction angles were examined for tempestuous liquid move through a round course with twin-rakish side mass infusion. For twin-side twin angular injections, an upstream distribution bubble shapes close to the hub and builds the size of a bubble by decreasing the primary and secondary injection angles. With a variation of 0,1 the bubble gains larger size. The speed of the centerline is at the top of the secondary injection field. The axial speed profiles have also been drawn at various axial locations which confirm the existence in the upstream primary injection site of a recirculation zone what's more, the completely evolved stream behind the half length of the canal.

REFERENCES

[1] Abdel-Fattah, A. (2007): Numerical and experimental study of turbulent impinging twin- jet flow, J. Experimental thermal and fluid science, **31**, pp. 1061-1072.

[2] Ashwin, G. and Chakraborty D. (2010): Numerical simulation of transverse side jet interaction with supersonic free stream. Aerospace Science and Technology, **14,** pp. 295-301.

[3] Bain D. B. and Smith C. E. (1992): CFD mixing analysis of jets injected from straight and slanted slots into confined cross flow in rectangular ducts, NASA, AIAA-92-3087.

[4] Giorges, A. T. G., Forney, L. J. and Wang X. (2001): Numerical study of multi jet mixing. Trans IChemE, **79**(A), pp. 515-522.

[5] Holderman, J. D. and Chang, C. T. (2008): Mixing of multiple jets with a confined subsonic cross flow Part III.

[6] Li, T., Pougatch, K., Salcudean, M. and Grecov, D. (2009): Numerical simulation of single and multiple gas jets in bubbling fluidized bed. Chemical Engineering Science, **64,** pp. 4884-4898.

[7] Moawad, A. K., Rajaratnam, N., Stanley, S. J. (2001): Mixing with multiple circular turbulent jets. Journal of Hydraulic Research, **39**(2), pp. 163-168.

[8] Patankar, S. V., 1981. Numerical heat transfer and fluid flow, McGraw Hill, New York.

[9] Tao, Y., Adler, W. and Specht, E. (2002): Numerical analysis of multiple jets discharging into a confined cylindrical crossflow, Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering, **216**(3), pp. 173-180.

[10] Tong-Miin L., Chin-Chun L., Shih-Hui C. and Hsin-Ming L. (1999): Study on Side-Jet Injection Near a Duct Entry With various Injection angles. J. Fluid Engineering, **121**, pp. 580-587.