Numerical simulation of twin turbulent side mass injection into a cross flow through a circular duct

SNEHAMOY MAJUMDER1, DEBAJIT SAHA2, PRITAM DAS3

1Associate Professor, 2Research Scholar, Department of Mechanical Engineering, Jadavpur University, Raja S. C. Mullick Road, Kolkata-700 032, West Bengal, INDIA
1srg_maj@yahoo.com, 2debajit.saha1986@gmail.com, 3bhargav_2006@yahoo.co.in

ABSTRACT

A numerical simulation has been carried out to study the effects of twin side mass injection with cross flow through a circular duct using modified $K - \varepsilon$ model, considering streamline curvature effects by modifying the model constants. $1/7$th turbulent velocity profile has been taken at the inlet. The ratio of the distance of the primary injection site from inlet to the length of the duct and the ratio between the axial distances between two injection slots to the length of the duct have been varied to visualize the change of the flow pattern in the circular duct. The width of both of the injection sites has also been gradually increased. With a single side mass entry through the wall, the momentum ratio of side mass velocity to inlet main velocity has been varied by changing the injection velocity as well as the inlet main velocity keeping the other parameters constant. It has been observed that, the flow pattern and the generation of recirculatory bubble and its changes in size depends on the momentum and velocity ratio of injection to inlet velocity irrespective to the velocity magnitudes at inlet and side injection sites.

Keywords: Twin side mass injection, $K - \varepsilon$ model, Streamline curvature, $1/7$th turbulent velocity profile, Momentum ratio of side mass velocity to inlet main velocity.

1. INTRODUCTION

Turbulent fluid flows in non-inertial frame of reference are encountered in a variety of engineering applications. In a gas turbine combustor, generally, the placements, size, number of injection sites and injection velocities of fluid greatly influences the performance of the combustor. Earlier studies of turbulent flow include the investigation on the fully developed turbulent flow by Laufer [1]. Experimental and numerical works have been done by several researchers for turbulent flow in porous ducts. These include the experimental studies of turbulent flow in a porous circular tube with uniform mass injection through the wall by Olson and Eckert [6]. Experiments have been carried out by changing the ratios of mass velocity from the tube wall to the velocity at the entrance cross section as well as taking zero entrance velocity at the upstream end of the tube. Other investigations included a porous channel of plane parallel walls with injection or suction varying exponentially with distance along the channel [2], porous tube with injection varying exponentially along the tube [3], a porous tube with uniform injection [4]. It has been observed from these analyses that component of fluid velocity normal to the main flow direction varied from a maximum velocity near the porous wall to minimum near the axis. Barcilon and Curtet [5] experimentally found that the Craya-Curtet number is an important parameter for characterizing the re-circulation zones that are often associated with mixing of jets. The numerical investigation of destabilization of the flow in a cylindrical duct has been reported by using the modification of standard model for streamline curvature by Launder and Spalding [7].

Tao et al. [9] numerically investigated the effects of a row of jets discharging normally into a confined cylindrical cross flow using the control volume based finite difference method. Ting-ting and Shao-hua [10] investigated interaction of turbulent jets with lateral injection into a cross flow. The results showed that the injection angle and jet to cross flow velocity ratio can change the flow fields and this change is comparatively high in the upstream side.

The two dimensional impinging circular twin jet flow with no cross flow was studied numerically and experimentally by Abdel-Fattah [11]. The jet Reynolds number, nozzle to plate spacing, nozzle to nozzle centre line spacing and jet angle has been varied and it is shown that the recirculation zone between twin jets becomes wider by increasing the above parameters. The increment of jet spreading decays by increasing the nozzle to plate spacing.

A numerical study to analyze the turbulent flow and heat transfer characteristics of the impinging slot jets in channel with cross flow has been carried out by Mushatat [13] with and without rib turbulators. The characteristics of air flow and heat transfer have been analyzed under different parameters and the obtained results shown that the recirculation regions, the local Nusselt number variation and the turbulent kinetic energy greatly affected by the size of the jets and ribs, rib thickness, distance between the jets and jet Reynolds number.
In the present study, turbulent fluid flow has been investigated with primary and secondary side mass injection through the wall of a circular duct. Momentum ratio of side injection velocity to main inlet velocity, P/L ratio, X/L ratio and Ds have been varied. Formation of recirculation bubble, its strength and size changes has been visualized from the flow contour and the parameters on which the structure of this bubble depends has been identified and analyzed. Standard $k - \varepsilon$ model together with modification due to streamline curvature has been employed to resolve the re-circulating flow regions accurately. The flow has been considered to be steady, incompressible, turbulent, non-reacting and axi-symmetric. The control volume formulation with power law scheme of S. V. Patankar [14] with SIMPLER algorithm has been adopted. The momentum and the $k - \varepsilon$ equations have been solved with the aid of wall function. The study is based on the solution of the complete Navier–Stokes equations and turbulence models using control volume formulation with power law scheme of S. V. Patankar. The tri-diagonal matrix algorithm (TDMA) is used to solve the discretization equations. The pressure–velocity coupling is achieved using the SIMPLER method. Two arrays of 251 X 151 and 301 X 201 grid points in axial and radial directions, respectively, have been used. It has been observed that the grid independent study has shown 0.001% change in the stream wise velocity change. The grid array of 251 X 151 has been used for all subsequent results reported here.

2. GEOMETRICAL DESCRIPTION

![Diagram](http://ijesr.in/)

Fig. 1, Geometry of the cylindrical duct with twin side mass injection site and turbulent flow at inlet

Figure 1 shows the essential features of the cylindrical axi-symmetric turbulent fluid flow, which has been considered in the present study with entries in the duct from both radial and axial directions. The cylindrical coordinate $r - X$ has been considered. Since the flow is axi-symmetric, so only the axis to the wall i.e., half of the circular duct has been considered for the numerical computation as shown in the figure 1. The inlet velocity is shown as $U_{in}$ and the radially inward velocity of side injection as $V_{inj}$. Number of injection sites, $n = 2$. The axial distance of the primary side injection site from inlet is denoted by $X_1$, while the diameter of the duct is given by $D$. The length of the duct is shown by $L$. The axial distance between the two injection sites is denoted by $P$. The injection width has been given by $D_s$. Here, the angle of side injection is 90°. Keeping this geometrical configuration the numerical investigation of turbulent fluid flow have been done by varying the P/L ratio, X/L ratio and Ds.

For the present investigation the axial length of the duct is considered as $L = 9$ m, the diameter of the duct, $D = 0.1534$ m, Air density, $\rho = 1.235$ kg/m$^3$. Molecular viscosity of air, $\mu_l = 1.853 \times 10^{-5}$ kgm$^{-1}$s$^{-1}$. The inlet velocity is given by, $U_{mean} = \left[1 - 2r/D\right]^{1/7}$ . The Reynolds number is defined as $Re = (\rho U_{mean} D)/\mu_l$, where $U_{mean}$ is the mass-averaged axial inlet velocity.

3. MATHEMATICAL MODELING

The mass and momentum conservation equations in axi-symmetric cylindrical co-ordinate system for the turbulent mean flow with eddy viscosity model is given as follows,

3.1 Continuity Equation

$$\frac{\partial (\rho u)}{\partial x} + \frac{1}{r} \frac{\partial (\rho v)}{\partial r} = 0 \tag{1}$$

3.2 Momentum Equations

3.2.1 Axial Component (X-Momentum)

$$\rho \left( \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial r} \right) = - \frac{\partial p}{\partial x} + \frac{\partial }{\partial x} \left( \mu_{eff} \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial }{\partial r} \left( r \mu_{eff} \frac{\partial u}{\partial r} \right) + \frac{\partial }{\partial x} \left( \mu_{eff} \frac{\partial u}{\partial x} \right) + \frac{1}{r} \frac{\partial }{\partial r} \left( r \mu_{eff} \frac{\partial v}{\partial x} \right) \tag{2}$$
3.2.2 Radial Component (r- Momentum)

\[ \rho \frac{\partial u}{\partial t} + \rho \frac{\partial u}{\partial r} \left( \frac{u}{r} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu_{\text{eff}} \frac{\partial u}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu_{\text{eff}} \frac{\partial v}{\partial r} \right) \]

The effective viscosity is, \( \mu_{\text{eff}} = \mu_l + \mu_t \)

The eddy viscosity is given by,

\[ \mu_t = \rho \mu \frac{k^2}{\varepsilon} \]

Modified \( C_\mu \) is given by,

\[ C_\mu = \frac{-K_1 K_2}{\left[ 1 + 8K_1 \frac{2}{\varepsilon} \left( \frac{S_U}{\sigma_r} + \frac{U_S}{R_C} \right) \right]} \]

Here, \( U_S = \sqrt{u^2 + v^2} \)

\( R_C \) is the radius of curvature of the concerned streamline (\( \psi \) = constant). To capture the streamline curvature effects the modifications has been incorporated according to Rodi and Leschziner [8] and implemented by Majumder and Sanyal [12]. These have been found effective to provide realistic solutions in the conditions of severe streamline curvature.

3.3. \( \kappa - \varepsilon \) Modelling

3.3.1. \( \kappa \)-Equation

\[ \rho \frac{\partial \kappa}{\partial t} + \rho \frac{\partial \kappa}{\partial r} \left( \frac{\kappa}{r} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( r \mu_{\text{eff}} \frac{\partial \kappa}{\partial r} \right) + \rho G - \rho \varepsilon \]

3.3.2. \( \varepsilon \)-Equation

\[ \rho \frac{\partial \varepsilon}{\partial t} + \rho \frac{\partial \varepsilon}{\partial r} \left( \frac{\varepsilon}{r} \right) = \frac{1}{\rho} \frac{\partial}{\partial r} \left( \rho \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial r} \right) + \frac{C_{\varepsilon 1} G \varepsilon}{\kappa} - C_{\varepsilon 2} \frac{\varepsilon^2}{\kappa} \]

Here, \( C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k, \) and \( \sigma_\varepsilon \) are the empirical turbulence constants, and some typical values of these constants in the standard \( \kappa - \varepsilon \) model are recommended by Lauder and Spalding [7] which are given as follows,

<table>
<thead>
<tr>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

The standard wall function has been adopted from Lauder and Spalding [7] for the solution of the \( k - \varepsilon \) equations for the problem investigated here.

4. VALIDATION OF THE PRESENT RESULTS WITH BENCHMARK SOLUTIONS

The present results have been validated with the benchmark solution [6]. Figure 2 presents the validation for the turbulent model used in the present analysis using the experimental results of Olson and Eckert [6] for a turbulent flow in a porous circular tube with uniform fluid injection throughout the tube wall.
Figure 2(a) and 2(b) shows the axial velocity profiles along the radius of the porous tube at X/D=14 and X/D=2 respectively with a velocity ratio of side mass injection to the inlet fluid flow, $V_{inj}/U_{in} = 0.0119$. The approaching flow is fully developed turbulent flow. $U$ is the mean temporal velocity in X-direction. The results include the experimental results of Olson and Eckert [6]. The matching of the experimental result with the present numerical method is quite good and the close agreement observed clearly validates the numerical scheme employed in the present study.

5. RESULTS AND DISCUSSIONS

5.1. Change of momentum ratio (Mr) by changing side injection velocity

The momentum ratio of side injection velocity to inlet main velocity has been varied by increasing the side injection velocity and keeping the inlet velocity constant. The flooded streamline contour along with the vector plot has been shown in fig. 3 for various momentum ratios. The side mass injection velocity gradually increased and it has been shown that with the change of this parameter the contour also change and at a certain injection velocity recirculation bubble appears in the flow region.

![Flow Visualization of Turbulent Fluid Flow in a Circular Duct at Various Momentum Ratio (V_{inj} Variation)](image)

When the velocity ratio is 1.1, the inlet main velocity can not overcome the high momentum exerts by the side mass flow and eventually there is a drop in pressure at the upstream which cuses the generation of recirculatory flow. With further increment of the velocity ratio, the length and breadth of the bubble also increases which gratefully affects the upstream flow by entrapping the upstream side by the formation of much bigger recirculation bubble. The width of the injection site and the distance of the injection site from the inlet has been kept constant.

![Axial Velocity Variation along the Radial Direction of the Duct](image)
The injection starts at a distance of 0.9 m from the inlet. The non-dimensional axial velocity profile along the radius of the duct has been shown at X = 0.9 m and at X = 2.0 m. From fig. 4(a), it is evident that up to a momentum ratio of 2.62, there is no negative value in velocity profile whereas, when the momentum ratio is 3.17, the profile tendency clearly supports the formation of recirculatory zone in the duct at an axial distance of X = 0.9 m. With the increase of the momentum ratio, the maximum negative value of axial velocity near the axis gradually increases, which is due to fact that as the injection velocity is much more in comparison with the inlet velocity, at upstream the velocity towards the reverse direction also increases gradually.

<table>
<thead>
<tr>
<th>Momentum Ratio (Mr)</th>
<th>Recirculation Length</th>
<th>Recirculation Width</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.656</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1.28</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2.12</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2.62</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3.17</td>
<td>0.103</td>
<td>0.0402</td>
</tr>
<tr>
<td>3.77</td>
<td>0.1833</td>
<td>0.0459</td>
</tr>
<tr>
<td>4.43</td>
<td>0.2532</td>
<td>0.0503</td>
</tr>
<tr>
<td>5.14</td>
<td>0.327</td>
<td>0.054</td>
</tr>
</tbody>
</table>

Table 1, shows the recirculation length and breadth at various momentum ratio of side mass velocity to inlet main velocity. Fig. 5(a) and Fig. 5(b) shows the variation of non-dimensional recirculation length and breadth with momentum ratio and it depicts from these figures that with the increase of momentum ratio the recirculation length and breadth both increases.

5.2. Change of momentum ratio (Mr) by changing inlet main velocity

From the above discussions, it is evident that if the momentum ratio increases by increasing the side mass injection velocity, at a certain momentum as well as velocity ratio recirculation bubble appears at upstream of the duct. Now, it is intended to study by keeping the side mass injection velocity a certain lower value and decreasing the inlet main velocity, whether the recirculation bubble appears at same momentum or not. Fig. 5 shows the contour of fluid flow where the inlet main velocity has been gradually decreases to change the momentum ratio of side injection velocity to inlet velocity. It has been observed that recirculation appears at a momentum ratio of 3.17 which exactly matches the results of fig. 1. And in this case also, with the increase of the momentum ratio the size of the bubble increases. By comparing fig. 3 and fig. 6, it can be said that, the flow pattern in the duct depends on the velocity as well as momentum ratio of side injection velocity to inlet main velocity, irrespective of the magnitude of inlet velocity and side injection velocity. The formation of recirculation bubble and its changes of pattern do not depend only on the value of injection velocity or inlet velocity rather it depends on the velocity and momentum ratio.

5.3. Variation of X1/L Ratio

The ratio of the axial distance of the primary injection site from the inlet (X1) to the length of the duct (L) has been varied and the change of flow pattern due to the change of this parameter has been analysed. As the gap between the two injection sites have been kept constant so, the secondary injection site also gradually shifted towards downstream of the duct. This ratio has been varied as X1/L = 0.1, 0.2, 0.3, 0.4. It has been observed that as the axial distance gradually increased i.e. if the injection sites shifted towards downstream, recirculation appears in the flow field and further shifting enlarges the size of the recirculation duct. This implies that the size of the recirculation zone appears in the flow field largely depends on the axial location of the injection sites as to get a bigger bubble, the injection ports have to shifted towards downstream when the number of injection sites is two. Though the flow with recirculation increases the mixing intensity and stabilizes the flame in a gas turbine combustor, the larger extent of recirculation may enhance the NOx emission phenomena which have the adverse affect on the combustion chamber. So, adequate location of the recirculation bubble and its size should maintain...
and it can be done by the proper location of the primary and secondary air injection sites in the combustion zone of gas turbine combustor.

Fig. 6, Flow Visualization of Turbulent Fluid Flow in a Circular Duct at Various Momentum Ratio ($U_{in}$ Variation)

The distribution of vorticity throughout the flow field has also been shown. It has been observed that at the wall near the injection zone and in between the injection sites the vorticity magnitude is quite large in comparison of the other portion of the duct as the radial velocity is more in this region due to the side mass injection through the wall of the duct. Near the axis, the vorticity magnitude is less and in the recirculation region it has negative value.

Fig. 8, Variation of Flow Pattern with the Variation of $X_1/L$ Ratio
5.4. Variation of Ds

In all the results discussed above, the injection width of both the injection sites has been kept as 0.1 m. In this section this parameter has been varied. The injection width has a vital role in the change of the momentum ratio of side mass velocity to inlet main velocity. The width has been varied as, Ds=0.1, 0.2, 0.3 and 0.4. The P/L ratio has been taken as 0.02, whereas, X1/L have been kept as 0.1. Keeping these parameters constant if the injection width gradually increased, then at the upstream recirculation bubble generated and with further increment of the injection width, the recirculation bubble size increases gradually. When the width is 0.2, the recirculation bubble generated near the axis, and when it increases to 0.3, the bubble size almost reaches up to the wall of the duct. When the bubble reaches up to the wall, there is a possibility of heating the wall of the duct, which is unlikely for the combustion chamber of a gas turbine combustor. When the injection width is 0.4, the bubble spreads in an irregular fashion over the upstream area of the duct and the recirculation bubble uplifts from the axis slightly. This irregular fashion increases with further increase of the injection width. It can be said that the upstream area mostly affected when the width is quite more. With the increase of the injection width, the mass of side injection fluid increases and the momentum of side injected fluid also increases eventually which in result affected the upstream area of the duct.

The vorticity magnitude near the injection area is more and it increases with the increase of the injection width. It has negative value near the recirculation region at upstream and when the width is 0.4, as the recirculation bubble spreads and thickens its size, the vorticity magnitude becomes positive at upstream again.

5.5. Variation of P/L Ratio

The ratio of the axial distance between the two injection sites to the length of the duct (P/L) has been varied and the effect of this variation on the flow field has been analysed. The side mass injection velocity has been kept constant for both of the injection sites. If the axial gap between the injections sites increases, the combined effect of side mass at upstream gradually decreases. The flow field between the primary and secondary injection zones greatly affected by the side mass injection and with the increase of P/L ratio, this affected flow field area increases. With the variation of this P/L ratio, recirculatory flow does not generated in the flow field. Except the zone between the two injection sites, the velocity field over the other region remains unchanged. The vorticity magnitude is large near the injection zone and wall, whereas, it is negative near the axis in between the injection zones.

6. CONCLUSIONS

The flow pattern and the formation of recirculation zone in the circular duct with side mass injection from twin side slots from the wall depends on Mr, P/L ratio, X1/L ratio and Ds. Recirculation bubble size increases with Mr, whatever the magnitude of Uin and Vinj is. With P/L ratio change, no recirculatory flow generated in the
flow field, whereas with the increase of \( \frac{X_1}{L} \) ratio and \( D_s \), recirculation bubble appears and its size increases with the variation of mentioned parameters. The recirculation generated near the axis and with the increase of \( \frac{X_1}{L} \) and \( D_s \) its size reached up to the wall of the duct. Vorticity magnitude is more in between the primary and secondary injection zones and near the wall throughout the duct. And it has negative value in the recirculation zone.

**Fig. 7. Variation of Flow Pattern with the Variation of P/L Ratio**

### NOMENCLATURE

- **C** \( \mu \) Empirical constant
- **C_1** Craya- Curtet number
- **D** Diameter of the cylindrical duct, m
- **D_s** Injection width, m
- **G** Production term
- **K_1, K_2** Constants
- **L** Length of the cylindrical duct, m
- **Mr** Momentum ratio of side mass flow to the main axial flow
- **P** Axial distance between the two injection zones, m
- **R** Radius of the cylindrical duct, m
- **r** Radial co-ordinate across the duct
- **Re_L** Recirculation Width, m
- **Re_W** Recirculation Depth, m
- **R_c** Radius of curvature of the streamline
- **\bar{u}** Time mean velocity along \( x \)-axis, m/s
- **U_{in}** Inlet flow velocity, m/s
- **U_{mean}** Mass-averaged mean axial velocity, m/s
- **\bar{v}** Time mean velocity along \( y \)-axis, m/s
- **V_{inj}** Side injection velocity, m/s
- **X** Axial co-ordinate along the duct, m
- **X_1** Axial distance of the primary injection zone from inlet, m
- **Re** Reynolds Number = \( \frac{\rho v D}{\mu} \)

### Greek Letters

- \( \varepsilon \) Turbulent dissipation rate, m\(^2\)/s\(^3\)
- \( k \) Turbulent kinetic energy, m\(^2\)/s\(^2\)
- \( \mu \) Viscosity of the air, kg/m-s
- \( \mu_{\text{eff}} \) Effective viscosity, kg/m-s
- \( \mu_l \) Laminar viscosity, kg/m-s
- \( \mu_t \) Eddy viscosity, kg/m-s
- \( \psi \) Stream function, m\(^2\)/s
- \( \rho \) Density of air, kg/m\(^3\)

### ACKNOWLEDGEMENTS

This work is supported by Council of Scientific and Industrial Research (CSIR), Govt. of INDIA.
REFERENCES