FLOW VISUALIZATION AND STUDY OF DRAG PARAMETERS WITH MASS BLOWING IN AXI-SYMMETRIC EXPANSION TYPE TURBULENT FLOW

Snehamoy Majumder, Arindam Mandal and Debjit Saha

Department of Mechanical Engineering, Jadavpur University, Kolkata, West Bengal, INDIA.

ABSTRACT
The turbulent fluid flow through sudden expansion passage has both fundamental scientific interest and numerous practical applications. The sudden change in the surface geometry of the passage increases the pressure along the direction of flow. Due to this adverse pressure gradient, the boundary layer separates at the sharp step edge and then forms a closed recirculation region containing the moving turbulent fluid. Due to the flow separation in the expanded space of the passage there is enhancement of the diffusion of the fluid in the downstream of the flow. This condition promotes the sustenance of flame in combustion chamber. The present analysis aims at studying the various aspects of the effects of inlet velocity imparted at the inlet of the sudden expansion passage to visualize the flow pattern and the changes of the size and strength of the recirculation bubble generated in the duct due to its sudden expansion with or without side injection. The analysis has been carried out by using modified $k-\varepsilon$ model, by modifying the model constants to incorporate the effects of streamline curvature.

Keywords: Sudden Expansion, Recirculation, Side Injection, Drag Parameter.

1. INTRODUCTION
Flows through sudden expansion are of interest from the point of view of fundamental fluid mechanics and numerous practical applications. There is keen interest in the understanding of such flows due to their wide spread occurrence in many fluid applications like pipe line, dump combustor, heat exchanger, nuclear reactor as well as Biological system. On the fundamental fluid mechanics side the flow through an axi-symmetric sudden expansion has all the complexities of internally separating and reattaching flows.

The turbulent axi-symmetric sudden expansion flow has been investigated both experimentally and numerically by many researchers. Laufer [1] carried out in detail the first experiment on the fully developed turbulent flow in a circular duct. Chaurvedi [2] first analyzed the flow characteristics in the axi-symmetric expansion by using the modified standard $k-\varepsilon$ model for streamline curvature. Lauder et al [3] numerically investigated the turbulent flow in a circular duct. Vasilev et al [4] numerically computed the turbulent flow in the sudden expansion of the channel. Smyth [5] experimentally examined the turbulent flow with separation and recirculation over the plane symmetric sudden expansion for the Reynolds number of 30,210. Szymocha [7] presented the experimental analysis for the turbulent water flow in the downstream of a plane symmetric sudden expansion. The investigation of the flow was carried out in a two-dimensional plane duct by employing the laser anemometer system. Gould et al [8] investigated the turbulent transport in the axi-symmetric sudden expansion. Chunbo et al [9] performed the numerical analysis on the Bingham turbulent flow in the straight circular pipe with sudden expansion. They observed that with the increase of the yield stress and plastic viscosity, the turbulent intensity decreases and the distribution of the turbulent intensity tends more towards the non-uniformity. Escudier et al [10] investigated the turbulent flow through the plane sudden expansion both experimentally and numerically. They studied the mean axial velocities, axial and transverse turbulence intensities and the Reynolds shear stress together with the wall pressure variation for an expansion ratio 4 at three spanwise and 13 axial locations. The numerical simulation was obtained by applying the $k-\varepsilon$ turbulent model. They observed that the maximum axial turbulence intensity occurred in the upper recirculation region. Such strong anisotropy of the Reynolds normal stresses was not seen in the lower recirculation region. By using modified $k-\varepsilon$ model over the two dimensional backward facing step geometry Mohanarangam et al [11] performed the numerical investigation for turbulent gas-particle flow, to study the effects of particle dispersion and its influence on step
heights. Lima et al [12] studied over a back ward facing step channel using two commercially available CFD code. They analyze three recirculation regions of the flow in a unilateral sudden expansion. A mathematical model was developed to simulate two-phase gas-dispersed flow moving through a pipe with axisymmetric sudden expansion by Terekhov et al [13]. In their model the two-fluid Euler approach was used. Fukagata et al [14] expressed the componential contributions that different dynamical effects make to the frictional drag in turbulent channel, pipe and plane boundary layer flows. They analyze the skin friction by decomposing four parts, i.e. laminar, turbulent as well as inhomogeneous and transient components. Direct numerical simulation of polymer-induced friction drag reduction in turbulent boundary layers describe by Dimitropoulos et al [15]. A clear idea of turbulent skin friction generated in flow along a cylinder given by Monte et al [16]. Majumder et al [17] investigate the side mass injection on turbulent flow with axial entry to a circular duct in the form of centrally confined jet. They predict that recirculation zone, friction factor, total turbulent energy, turbulent shear stress, Taylor scale Reynolds number and turbulent energy flux show gradual reduction with increase in the side injection velocity.

2. GEOMETRICAL DESCRIPTION

In Fig 1 the flow geometry has been shown in the cylindrical coordinate system with r-x axis. The flow at the inlet is non-reacting and along the axial direction. In the fig. the velocity inlet is shown by \( U_{in} \) and injection velocity is shown by \( V_{inj} \). The value of the inlet velocity varies from 0.5 m/s to 8.0 m/s. Injection velocity is constant i.e. 0.01 m/s.

![Schematic diagram of the axi-symmetric sudden expansion.](image)

The upstream radius and downstream radius are 2.5 m and 3.0 m respectively. The upstream length of the passage is 5 m and downstream length of the passage is 30 m.

3. BOUNDARY CONDITION

3.1 At the Inlet

(a) The inlet axial velocity is uniform. i.e., \( U_{in}=\text{Constant} \)

(b) The tangential velocity (If present at all).

(c) Turbulence energy and dissipation rate are taken as:

\[ K_{in}=0.003U_{in}^2 \]

\[ \varepsilon = \frac{c_{\mu}K_{in}^{3/2}}{0.003R} \]

3.2 At the Wall

No slip wall boundary condition has been taken for the solid wall.

i.e., \( \frac{\partial \phi}{\partial r} = \frac{\partial \phi}{\partial r} = u = v = w=0 \) (Without Side Injection)

\( v = -U_{inj} \) With slip condition of \( \frac{du}{dr} = 0 \)

3.3 At the Axis

Zero shear stress condition has been taken for the axis.

i.e., \( \frac{\partial \phi}{\partial r} = 0 \)

Where \( \phi = u, v, w, k, \varepsilon \)

3.4 At the Outlet

Fully developed flow condition has been taken at the outlet.

i.e., \( \frac{\partial \phi}{\partial r} = 0 \)

4. GOVERNING EQUATIONS

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

Continuity equation:

\[ \frac{\partial (\rho U)}{\partial x} + \frac{1}{r} \frac{\partial (\rho r U)}{\partial r} = 0 \]

Axial Component (x-component):

\[ \left[ \frac{r}{2} \frac{\partial (\rho U)}{\partial r} + \frac{U}{2} \frac{\partial \rho}{\partial r} \right] = -\rho \frac{\partial U}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u'' \frac{\partial U}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u'' \frac{\partial \rho}{\partial r} \right) \]

Radial Component (r-component):

\[ \rho \left[ \frac{r}{2} \frac{\partial V}{\partial r} + \frac{V}{2} \frac{\partial \rho}{\partial r} \right] = -\rho \frac{\partial V}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u'' \frac{\partial V}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial r} \left( \rho u'' \frac{\partial \rho}{\partial r} \right) \]

The effective viscosity is given by,

\[ \mu_{eff} = \mu_t + \mu_k \]

The eddy viscosity is given by,

\[ \mu_k = \rho c_{\mu} k^2 / \varepsilon \]
given by
\[ C\mu = \frac{-K_1K_2}{1 + 8K_1^2\frac{K_2}{\varepsilon^2}\left(\frac{\partial U_S}{\partial \eta} + \frac{U_S}{R_C}\right)\frac{U_S}{R_C}} \] (6)

In the equation (6) \( U_S = \sqrt{u^2 + v^2} \), \( R_C \) is the radius of curvature of the streamline concerned (\( \psi \) constant). The value of \( K_1 \) and \( K_2 \) are taken as 0.27 and 0.3334 respectively.

Turbulent Modeling:

\( k \) and \( \varepsilon \) can be obtained from the Navier-Stokes equation. The \( k \) and \( \varepsilon \) equations are as follows,

\[ k - \text{Equation:} \]
\[ \rho \left[ \frac{\partial k}{\partial t} + \frac{\partial (\mu_1 + \mu_2)}{\partial x} + \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial k}{\partial r} \right) \right] = \rho G - \rho \varepsilon \] (7)

Where \( G \) is the production term and is given by
\[ G = \mu_1 \left[ 2 \left( \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \sigma \left( \frac{\partial v}{\partial z} \right)^2 \right] \] (8)

\[ \varepsilon - \text{Equation:} \]
\[ \rho \left[ \frac{\partial \varepsilon}{\partial t} + \frac{\partial (\mu_1 + \mu_2)}{\partial x} + \frac{1}{r} \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial \varepsilon}{\partial r} \right) \right] + C_{S1} \frac{\varepsilon}{k} - C_{S2} \frac{\varepsilon^2}{k} \] (9)

Here \( C_{\mu}, C_{S1}, C_{S2}, \sigma, \sigma_k \) are the empirical turbulent constant. The values are considered according to Launder et al. [3], 1974 and the values are 0.09, 1.44, 1.92, 1.0 and 1.3 respectively.

Coefficient of friction:
\[ C_f = \frac{\tau_w}{\rho U_{av} v} \] (10)

Here \( C_f \) is coefficient of friction, \( \rho \) is density of the fluid and \( U_{av} \) is average inlet velocity. Where \( \tau_w \) is shear stress.
\[ \tau_w = \mu \frac{du}{dy} \] (11)

Where, \( \mu \) is the fluid viscosity.

5. RESULT AND DISCUSSIONS

For analyzing the effects of the Reynolds number on the recirculation bubble, the Reynolds number has been varied by varying the axial inlet mean velocity and keeping geometry of the passage constant.

The inlet velocity is uniform and denoted by \( U_m \) as shown in the Fig 2 with increase of \( U_m \) the Reynolds number increases and the Reynolds number is varied by changing the inlet velocity from 0.5 m/s to 8 m/s. Here geometry and expansion ratio of the geometry kept remain same as expansion ratio is 1.2.

In the first Plot of Figures 2, 3, 4 inlet velocity consider as 0.5m/s and corresponding Reynolds number (Re) 84.128×10³. The recirculation bubble is generated due to the expansion of the passage and the incoming fluid could not negotiate the expansion passage wall all along completely and finally diffused to generate the secondary flow. The recirculation bubble generated is sheltered at the corner expansion of the passage. This is the general feature of the sudden expansion flow. The strength and reattachment length of the recirculation bubble are dependent on the geometry of the passage in absence of any change in boundary conditions. From the vector diagram we also see the passage properly.

Fig 2. Effect of recirculation size with the variation of inlet velocity (Streamline Plot)

In the second plot of those three figure axial inlet velocity is increased to 1 m/s keeping all other parameters same as that of the previous case. The corresponding Reynolds number for the relevant inlet velocity is 1.6825×10⁶. From the 2nd plot of the figure 2 it is observed that the dividing streamline and the reattachment point shifted toward the corner portion of the passage with the increase in the inlet velocity. The recirculation bubble generated in the second case is of smaller size than the first case. This means the strength of the recirculation bubble in the second case is less than the first case. The direction of the flow in the recirculation zone remains in the anti-clockwise direction.

In the third plot of the above figures the inlet velocity is increased to 2 m/s, keeping all other parameters same as that described in the previous cases. The Reynolds number for this inlet velocity is 3.365×10⁶. From the streamline plot of the above figure it can be observed that the dividing streamline moves...
further toward the corner portion of the passage and thereby decreasing the size and strength of the recirculation bubble with respect to the earlier two cases. Here also the direction of the flow in the recirculation regions remains same as that of the previous cases.

Fig 3. Effect of recirculation size with the variation of inlet velocity (Vector Plot)

In the fourth and fifth case the inlet velocity increase to 4m/s and 8m/s and the Reynolds numbers 6.73×10^6 and 13.46×10^6 respectively. In both cases we observe that stream line moves more towards corner point with the respective previous cases. The recirculation size and strength is also less.

Fig 4. Effect of recirculation size with the variation of inlet velocity (Flooded Contour Plot)

In the figure 5 the variation of the recirculation length with respect to the Reynolds number has been plotted. From the above figure it is observed that the recirculation bubble length or reattachment length decreases with the increase in the Reynolds number. The figure also confirms the fact that though the recirculation bubble strength is reduced but the same is not eliminated.

A correlation has been obtained between the Reattachment length and Reynolds number as given below:

\[
\text{Reattachment Length} = -2e^{-0.8\text{Re}} + 3.340
\]

From this equation it is concluded that recirculation length is decrease correspondingly due to increase of Reynolds number.

Fig 6. Effect of Coefficient of friction due to variation of Re

In figure 6 we see the variation of coefficient of friction due to change in Re. Here we see that in the downstream the value of C_f correspondingly decreases. As the average velocity of the flow in the inlet is increasing and this causes decrease in the probability of the generation of adverse pressure gradient which is reasonable for this. Here also see the chaotic situation just after upstream zone. It is may be the effect of recirculation.

Fig 5. Variation of Reattachment length with respect of Reynolds number.

In figure 7 we see the effect of mass blowing and as
a consequence the recirculation length quite smaller from the previous case.

Fig 8. Variation of turbulent Shear stress in mass blowing.

In figure 8 the radial distributions of turbulent shear stress in mass blowing have been plotted. In this case it is observed that the maximum turbulent shear stress generates in the central part of the recirculation zone. After that the stress is decreasing trend with the radius.

6. CONCLUSION

It has been observed that in a sudden expansion the corner reattachment length and breadth or the recirculation bubble strength decrease with the increase of the Reynolds number. It has been also observed that the value of the friction coefficient $C_f$ decreases with the increase of the Reynolds number in the downstream of the reattachment point. However the friction coefficient has a mixed behavior within the recirculation site, i.e. initially it increases and then it gradually decreases.

7. REFERENCES


8. NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_{st}$</td>
<td>Empirical Constant</td>
<td>-</td>
</tr>
<tr>
<td>$C_{t}$</td>
<td>Empirical Constant</td>
<td>-</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------------------------------------------------------------------------------------------------</td>
<td>--------------------</td>
</tr>
<tr>
<td>( \bar{C}_u )</td>
<td>Empirical Constant</td>
<td>-</td>
</tr>
<tr>
<td>( \bar{u} )</td>
<td>Time mean velocity along Z axis.</td>
<td>(m/s)</td>
</tr>
<tr>
<td>( \bar{p} )</td>
<td>Time mean velocity along r axis</td>
<td>(m/s)</td>
</tr>
<tr>
<td>( u_{in} )</td>
<td>Average inlet velocity</td>
<td>(m/s)</td>
</tr>
<tr>
<td>( E_r )</td>
<td>Expansion Ratio ( R/R_i )</td>
<td>-</td>
</tr>
<tr>
<td>( G )</td>
<td>Rate of Production</td>
<td>-</td>
</tr>
<tr>
<td>( k )</td>
<td>Turbulent kinetic energy</td>
<td>(m(^2)/s(^2))</td>
</tr>
<tr>
<td>( r )</td>
<td>Radial co-ordinate across the duct</td>
<td>-</td>
</tr>
<tr>
<td>( L )</td>
<td>Length at the down stream</td>
<td>(m)</td>
</tr>
<tr>
<td>( L_i )</td>
<td>Length at the upstream</td>
<td>(m)</td>
</tr>
<tr>
<td>( L_R )</td>
<td>Reattachment length</td>
<td>(m)</td>
</tr>
<tr>
<td>( r_i )</td>
<td>Radius at inlet</td>
<td>(m)</td>
</tr>
<tr>
<td>( Re )</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>( Z )</td>
<td>Axial co-ordinate along the duct</td>
<td>-</td>
</tr>
<tr>
<td>( \mu_l )</td>
<td>Molecular or laminar viscosity</td>
<td>(N-s/m(^2))</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>Turbulent viscosity</td>
<td>(N-s/m(^2))</td>
</tr>
<tr>
<td>( \mu_{eff} )</td>
<td>Effective viscosity</td>
<td>(N-s/m(^2))</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>Turbulence kinetic energy dissipation rate</td>
<td>(m(^3)/s(^3))</td>
</tr>
<tr>
<td>( \sigma_k )</td>
<td>Prandtl number of the turbulent kinetic energy</td>
<td>-</td>
</tr>
<tr>
<td>( \sigma_\varepsilon )</td>
<td>Dissipation Energy</td>
<td>-</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density</td>
<td>(Kg/m(^3))</td>
</tr>
</tbody>
</table>

8. MAILING ADDRESS

Dr. Snehamoy Majumder  
Associate Professor  
Department of Mechanical Engineering  
Jadavpur University, Kolkata-700032,  
WEST BENGAL, INDIA  
Phone: +919831369134  
E-mail: srg_maj@yahoo.com