Cfd Simulation Of Flow Field In An Inject Pipe Mixer
K. S. Sudeep and S. Murthy Shekhar
School of Biotechnology, Chemical and Biomedical Engineering
VIT University, Vellore -632014

The Inject Pipe Mixer (IPM) are being used as venturi injectors and can also homogenize two fluids. The entry of secondary fluid as a jet influences flow field in the device. A CFD simulation of flow through an IPM (L-shape) has been carried out using ANSYS CFX 11.0 using water as the flowing fluid to understand the flow field under turbulent flow conditions using standard k-ε model. The results of simulations indicate flow field dependence on both main pipe velocity and side pipe inlet velocity. The flow field depends on V and W components and U component contributes very marginally. The velocity magnitude depends on location inside the geometry.

1. Introduction

Inject Pipe Mixer (IPM) is used to homogenize same or different fluids either at same or different temperatures. It can also be used when tracer elements or additives need to be introduced into the main flow stream. IPM is a simple energy conservation device which can enhance heat transfer and fluid mixing, without any external device. These devices are simple in construction, fabrication and also application. A typical sketch of an IPM is shown in Fig. 1. The performance of an IPM depends on material of construction, geometry and flow field. For a given material and geometry, flow field depends on the type of jet which in turn depends on diameter of main and side pipe, point of attachment of side pipe on to the main pipe, nature of the fluid, velocity of the fluid, operating pressure and temperature and surface roughness of the pipe. Although simple in design and construction, little is known about inside flow field and performance.

A literature of relevance to present work, was the study of Hosseini S.M. et al. (2008). The above study was based on usage of Particle Image Velocimetry (PIV) technique to understand heat transfer mechanism. In their study, Hosseini et al. (2008) have reported flow inside an IPM dependence on curvature ratio of the bend, axial distance between the bend and T-junction area.

2. CFD simulation

In the present work a three dimensional model of the Inject Pipe Mixer shown in Fig. 1 has been simulated. The above geometry has been created and meshed using ANSYS Workbench and CFX Mesh Module respectively. The simulations have been carried using a finite volume based commercial CFD code ANSYS CFX 11.0 developed by ANSYS Inc., USA respectively. The details of the geometry have been given in Table -
1. The major regions defined were main inlet, main outlet, and side pipe inlet. All other regions have been set as wall.

![Fig. 1 Schematic Diagram of IPM](image)

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Main Pipe Diameter (m)</td>
<td>0.051</td>
</tr>
<tr>
<td>Side Inlet Diameter (m)</td>
<td>0.021</td>
</tr>
<tr>
<td>Length of Main pipe (m)</td>
<td>2.0</td>
</tr>
<tr>
<td>Height of Main Pipe (m)</td>
<td>2.0</td>
</tr>
<tr>
<td>Height of side pipe (m)</td>
<td>0.3</td>
</tr>
<tr>
<td>Radius of Curvature</td>
<td>45°</td>
</tr>
</tbody>
</table>

To obtain the required flow field, solutions have been obtained by solving the basic governing equations viz., continuity equation and momentum equations with inlet velocity normal to main inlet and side inlet as boundary conditions or input parameters. Further, no slip condition was imposed at the walls. The characteristics of flowing fluid was considered to represent water by imposing a density and viscosity value of 997.0 kg/m$^3$ and 0.001 Pa.s. All the simulations have been carried out assuming steady state conditions for velocities varying from 1 to 3 m/s. With the above velocities, Reynolds number was found to vary from 50,847 to 1,52,541 indicating turbulent flow. Hence, all the simulations have been carried using Standard K-ε model. The solutions have been obtained imposing coupled solver approach using advection scheme with high resolution and a convergence limit of 1 x 10$^{-5}$.

3. Results

3.1 Grid independence and validation

The accuracy of the CFD simulation results depends on stability, consistency and convergence criteria envisaged in the simulations. The accuracy of simulations also depends on the number of grid elements used in the simulations. Hence, simulations have been carried out using mesh volumes varying from 178724 to 187689 by varying body spacing on the geometry from 0.12 to 0.06, wherein decrease in body spacing value leads to increase of number of mesh volumes. The grid independence study and validation have been carried out at main and side pipe inlet velocity of 3 m/s, which represents highest Reynolds number for the present study. The simulated results have been verified for variation in velocity magnitude of fluid stream across the radial direction, at various locations viz., Line 1: 1.0 m from inlet, Line 2: 0.0 m normal to the point of injection and Line 3: 1.0 m below the outlet. The variation in magnitude of velocity at Line 1 has been shown in Fig. 2. The figure indicate, only marginal change in the velocity profile with increase in mesh volumes, indicating number of mesh volumes are used in the present study are sufficient to predict the flow profile. A similar agreement in results have been observed even at Line 2 and Line 3 respectively. Hence, further simulations have been carried out at total mesh volume of 1,85,653 representing a body spacing of 0.08 in order to minimize computational time.
Further the grid independent results have been validated by comparing the velocity profile obtained at Line 1, with the Prandtl’s 1/7 power law relation valid for prediction of velocity profile for turbulent flow. A comparison of the CFD results and the correlation are shown in Fig. 3. The figure shows a good agreement of results in the core central region, and small deviation near the wall. The small deviation is due to weakness of the Prandtl’s correlation in estimating velocities in the near wall region. The good agreement of results, validate the procedure envisaged to predict the flow field in the IPM using CFD.

3.2 Flow Field

With reasonable grid independence and validation results, velocity distribution viz., velocity magnitude, U, V and W components along radial direction at Line 1, Line 2 and Line 3 are shown in Fig. 4. Fig. 4(a) representing velocity distribution at Line 1, shows W velocity component contributes substantially to the velocity magnitude when flow is along the horizontal direction, while contribution of U and V components are small in magnitude and nearly small in comparison with W component. Fig. 4(b) representing the velocity distribution at Line 2, indicates an higher magnitude of velocity due to the entry of fluid from side pipe. A higher degree of variation exists in the zone of side pipe fluid entry leading to absence of symmetry in flow distribution found at Line 1. The velocity component contributing to the velocity magnitude has swapped from W to V component. The radius of curvature has influenced the flow distribution and contribution of different components to velocity magnitude. Further it can be seen that there is no change in the U component velocity distribution across the cross section. From Fig. 4(c) of Line 3, it can be seen that, the velocity magnitude across the cross section has approached a constant value of 3.5 compared to 3.1 for Line 1. Again, significant change is the shift of major contribution for velocity magnitude to V velocity component in contrast to W velocity component at Line 1. The flow distribution of U and W velocity component exhibit symmetry but are of small magnitude. From visualization of all the above plots, it can be inferred that, contribution of U velocity component is very small but overall distribution pattern
depends on location. A vector plot obtained from the simulated results showed streamlined pattern with magnitude of vector varying along the flow direction depending on the location in the IPM.

![Vector plot](image)

**Fig. 4a Velocity magnitude and velocity components at Line 1**

![Velocity profile](image)

**Fig. 4b Velocity magnitude and velocity components at Line 2**

![Velocity profile](image)

**Fig. 4c Velocity Magnitude and velocity components at Line 3**

### 3.4 Effect of Main Stream Fluid Velocity

The performance of IMP has been verified for effect of main pipe velocity between 1 - 3 m/s and side pipe velocity of 3 m/s. The results of flow profile in terms of velocity magnitude at Line 1 and Line 2 are shown in Fig.5a and5b respectively. Fig. 5a shows increase of velocity magnitude with increase in inlet velocity and velocity distribution symmetric across the cross section. The flat distribution over the cross section except near the walls resembles a typical turbulent flow profile. But Fig. 5b shows, velocity magnitude increasing with increase in velocity and asymmetric velocity distribution due to addition of fluid from side pipe. The influence of side pipe velocity has been found to decrease with increase in main pipe velocity. Further it has been observed that, at Line 3 (figure not shown) velocity distribution has been found to exhibit trend similar
to that at Line 1, but with a higher magnitude profiles due to addition of fluid from side pipe.

Further analysis of components of velocities indicated that, the magnitude of $U$ component velocity to vary from $-1 \times 10^{-4}$ to 0.0 with increase in input velocity. The distribution of $V$ component at Line 2 has been shown in Fig. 6. The figure shows, $V$ component velocity increasing with increase in main velocity. A comparison with Fig. 5b shows, magnitude of $V$ component is very close to the velocity magnitude values indicating it as a major contributor to velocity magnitude at Line 2. The peak values velocities in between the radial distance of 0.49 to 0.51 were found to be due to side inlet fluid velocity. The distribution of $W$ velocity component at Line 2 is shown in Fig. 7. It shows, the $W$ velocity contribution to the magnitude of velocity decreases with curvature transferring the kinetic energy to the $V$ velocity component.
3.5 Effect of side stream velocity
The effect of side inlet velocity has been verified by varying for side inlet velocity from 0 – 3 m/s for different main inlet velocities. The side inlet velocity of 0.0 m/s represent flow in a bend. The variation in velocity magnitude along radial distance at Line 2 is shown in Fig. 8. The velocity magnitude shows increase in value with increase in side stream velocities. But no appreciable change in the flow pattern, except in the zone of the side inlet fluid flow.

4. Conclusions
From the study undertaken for determination of flow field of an IPM, it can be concluded that, CFD technique can be used for understanding about the performance of IPM. The flow field depends on the main inlet velocity as well as side inlet velocity. The flow distribution is mainly function of V and W velocity with 90° bend. The flow distribution also depends on the percentage contribution of side inlet flow to the main flow. Further investigation about the effect of temperature and roughness are being investigated.

5. References