Optimum diffuser geometry for the automotive catalytic converter

G S Kulkarni, S N Singh, V Seshadri & Ratan Mohan

Department of Applied Mechanics, Indian Institute of Technology, New Delhi, India
Department of Chemical Engineering, Indian Institute of Technology, New Delhi, India

Received 8 March 2002; accepted 15 November 2002

Catalytic converters have become an integral part of modern cars as they provide excellent control on pollutants but also lead to reduction in the overall efficiency of the engine. In the present study, the geometry of the inlet header, which is primarily a diffuser, has been optimized using Computational Fluid Dynamics (CFD) for minimizing the pressure loss. It has been found that the optimum design of the diffuser is the one having moderate diffuser half angle for most part of its length and then gradual but steeper diffusion in the later part along with a smooth wall curvature.

Air pollution is an identified problem the world over and vehicular pollution accounts for more than 50% of the total air pollution. In Delhi, it accounts for 70% of the total air pollution. Pollutants generally present in auto emissions are carbon monoxide, unburnt hydrocarbons, oxides of nitrogen and particulate matter. All modern cars are provided with catalytic converters to control these emission levels to the minimum. Emissions (g/km) for 1.8 L petrol engine with closed loop fuel injection are: No catalyst (Co 5.99, HC 1.67, NOx 1.04), with catalyst (Co 0.61, HC 0.07, NOx 0.04), and for 1.9 L diesel engine with direct injection are: No catalyst (Co 1.20, HC 0.38, NOx 0.54, particulate 0.07), with catalyst (Co 0.17, HC 0.05, NOx 0.42, particulate 0.04), [Courtesy: Johnson Matthey ply, London].

These devices are generally placed near to the engine exhaust as its operation depends on the emission heat to boost the catalytic reactions. Though the catalytic converters provide excellent control on pollutants in the auto emission, these lead to reduction in the overall efficiency of the engine due to the increased resistance to flow in the exhaust system and hence, result in the reduction of the achievable peak power. A Catalytic converter is a device, which is fitted in the exhaust system of an automobile engine. It consists of inlet pipe, inlet header generally in shape of a diffuser, plenum-chamber, contraction nozzle and exhaust pipe. The plenum chamber has a metallic honeycomb substrate, which is formed by rolling a corrugated thin foil with a plain foil. The plain foil is coated with a very thin wash coat of precious metals namely platinum, palladium and rhodium. The present catalytic converters provide an excellent emission reduction but limit the peak power, which in turn affects the fuel economy. To overcome this shortcoming, there is need to optimize the design. Presently it seems that no major changes are possible in the plenum chamber but design changes are feasible in the inlet header.

Inlet header is a transition piece in the shape of a diffuser, which connects the inlet pipe to the plenum chamber. Design of this transition piece is very crucial as it strongly affects the pressure loss due to fluid friction and the flow maldistribution as a result of flow separation. The input parameters for the design of the inlet header are area ratio and its length. There is a need to optimize the shape of the diffusers to improve their performance which in turn will reduce the peak power loss.

Daniel et al. have investigated the restriction characteristics of monolith converters and found that header losses account for 50% of the overall loss. They have also shown that off-setting the exit pipe results in higher losses. Kim et al. have made numerical investigation to establish the trade off between flow uniformity and pressure drop in an axisymmetric catalytic converter. Similar investigations have been reported for monolith automotive catalytic converter by Lai et al. using the CFD code named 'Phoenics'. Paul and Sacha have analyzed the substrate factors which influence the pressure drop and conversion efficiency of the catalyst system. Barris developed a design based on the optimization of combined catalytic converter-muffler considering the effect of the substrate, the
total system back pressure and its acoustical behavior for low emission diesel engines. Daniel et al.\(^7\) have suggested a diffuser design for minimum pressure loss. The suggested diffuser geometry involves boundary layer separation but diffuses the flow to larger diameters with better flow distribution and lower pressure losses. This geometry is identified as EDH geometry in the present study.

The relevant literature survey shows that there is substantial scope to reduce the pressure losses in the catalytic converter by improving the diffuser design of the inlet header. In the present study, an attempt has been made to suggest different diffuser designs for improved performance of the catalytic converter in terms of pressure loss based on the concept of overall diffuser effectiveness elaborated extensively by Sovran and Klomp\(^8\). Diffusers with outlet pipes/downstream elements are not affected by the inlet conditions as much as similar diffusers with free discharge. For catalytic converters, where diffuser has an outlet pipe/downstream elements and the area ratio exceeds the normal prescribed limits, it is better to reduce the area ratio in the initial length and then have a sudden expansion to feed the flow to the outlet pipe/down-stream element. It is expected that this combination will result in efficient diffusion.

**Computational Procedure**

A computer code developed by Agrawal et al.\(^9\) for prediction of two dimensional (plane or axisymmetric) flows in the arbitrary geometries has been adopted and modified to carry out the present study. The basic laws of fluid flow are the conservation of mass and momentum and can be described in coordinate free form for steady flow as:

\[
div(\rho \vec{U}) = 0 \quad \text{... (1)}
\]

\[
div(\rho \vec{U} \vec{U} - \bar{T}) = \bar{S}_U \quad \text{... (2)}
\]

According to Demirdzic\(^10\), these equations for steady, incompressible subsonic flow can be expressed in general Cartesian co-ordinate as:

\[
\frac{\Delta}{\Delta x_j}(\rho \bar{U}_j \alpha_{ij}) = 0 \quad \text{... (3)}
\]

\[
\frac{\Delta}{\Delta x_j} \left[ (\rho \bar{U}_j \bar{U}_j - \tau_{ij}) \alpha_{ij} \right] = \bar{S}_{\alpha_{ij}} \quad \text{... (4)}
\]

\(\tau_{ij}\) is the stress tensor and is expressed as:

\[
\tau_{ij} = -p \delta_{ij} + \mu \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right] \quad \text{... (5)}
\]

where \(\delta_{ij}\) is the Kronecker's delta.

In turbulent flows, small-scale high frequency fluctuations are always present. To account for these fluctuations, a time averaging procedure given by Demirdzic\(^10\) is employed to express the equations as:

\[
\frac{\Delta}{\Delta x_j}(\rho \bar{U}_j \alpha_{ij}) = 0 \quad \text{... (6)}
\]

\[
\frac{\Delta}{\Delta x_j} \left[ (\rho \bar{U}_j \bar{U}_j - (\bar{\tau}_{ij} - \rho \bar{U}_i \bar{U}_j)) \alpha_{ij} \right] = \bar{S}_{\alpha_{ij}} \quad \text{... (7)}
\]

Here \(-\rho \bar{U}_i \bar{U}_j\) are the Reynolds stresses and these need to be modeled in order to get closure solution. In the present study, this has been achieved by using two-equation k-\(\varepsilon\) turbulence model. Reynolds stresses are expressed in terms of turbulent viscosity and velocity gradients as:

\[
\gamma_{ij} = -\rho \bar{U}_i \bar{U}_j = \mu_i \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right] \quad \text{... (8)}
\]

In the above equation \(\mu_i\) is related to the turbulent kinetic energy 'k' and dissipation rate '\(\varepsilon\)' as:

\[
\mu_i = C_u \rho \frac{k^2}{\varepsilon} \quad \text{... (9)}
\]

The transport equations for \(k\) and \(\varepsilon\) for steady flow have been given by Launder and Spalding\(^11\) in general variable '\(\phi\)' as:

\[
div(\rho \bar{U} \phi - \bar{q}_\phi) = S_\phi \quad \text{... (10)}
\]

and in Cartesian co-ordinates as:

\[
\frac{\Delta}{\Delta x_j} \left[ (\rho \bar{U}_j \phi - q_{\phi j}) \alpha_{ij} \right] = S_\phi \quad \text{... (11)}
\]
In this equation, $q_{\phi i}$ is the turbulent flux for a scalar quantity and is expressed as:

$$q_{\phi i} = \Gamma_\phi \frac{\partial \phi}{\partial y_i} \quad \ldots (12)$$

where $\Gamma_\phi$ represents turbulent diffusivity of the scalar quantity and is written as:

$$\Gamma_\phi = \frac{\mu_1}{\sigma_\phi} \quad \ldots (13)$$

Here $\sigma_\phi$ is a constant. The source terms of $'k'$ and $'\varepsilon'$ are:

$$S_k = G - \rho \varepsilon \quad \ldots (14)$$

and,

$$S_\varepsilon = C_1 \frac{\varepsilon^2}{k} - C_2 \rho \varepsilon \quad \ldots (15)$$

The function $G$ is given as:

$$G = \Gamma_{ij} \frac{\partial U_j}{\partial x_i} \quad \ldots (16)$$

In these equations, a set of constants $C_1$, $C_2$, $\sigma_\phi$, and $\sigma_\varepsilon$ are empirical constants and have the values of 0.09, 1.44, 1.92, 1.0 and 1.33 respectively.

The solution of the governing equations for real life flows is obtained using numerical technique as the equations are non linear and coupled. Accuracy and stability of the numerical solution depends on the method adopted for the discretization. The present numerical scheme is based on the finite volume approach. The details of the approach are explained elsewhere. A simple non-orthogonal 2D algebraic grid (Fig. 1a) is employed in the selected geometries. After grids, the governing equations are discretized using finite volume approach. The final form for the discretized equation including all fluxes components is written as:

$$[A]\{\phi\} = \{S\} \quad \ldots (17)$$

where $[A]$ is a $M \times M$ coefficient matrix. $M$ is the total number of control volumes in the grid; $\{\phi\}$ is the dependent variable vector of $M$ nodal values, $\phi$, $U$, $V$, $P$, $K$, etc.; $\{S\}$ is a vector of $M$ source terms. The above set of equations is solved using the strongly implicit procedure. For an iterative solution, the above equations are written as:

$$a_{ij} \phi_j = \sum_{nb} a_{ij} \phi_{nb} + S_j \quad \ldots (18)$$

where $\phi$ is the dependent variable, nb are the neighboring nodes ($E$, $W$, $N$, $S$; Fig. 1b) and $a_{ij}$ is $A_{ij}$.

To ensure convergence, under relaxation is employed.
Boundary conditions

Complete specifications of the inlet, exit and wall boundary conditions associated with the problem are specified to solve the governing equations. These are of two types namely Dirichlet type in which the boundary values like velocity, etc., are specified and Von Neumann type where in boundary fluxes, etc., are specified.

Validation tests

The computer algorithm developed has been validated against Nikuradse's profile for pipe flow. Flat velocity profile was fed at the inlet of pipe having overall length more than 60 diameters. The predicted velocity profile at 60 diameters has been compared with the measured turbulent velocity profile and both agree reasonably well. The second validation test has been done for flow in a conical diffuser. The predicted results at different locations along the length compare reasonably well with experimental results of Habib and Whitelaw\(^4\). The deviations observed were within ±5% for the velocity profile at the outlet of the conical diffuser (Fig. 2).

Geometries of investigated catalytic converter

Fig. 3 shows the physical geometry of a catalytic converter assembly, which consists of inlet pipe, diffuser, 1\(^{st}\) and 2\(^{nd}\) monolith with air gap, contraction or nozzle and exit pipe. In the present study, the focus is on improving the design of the inlet diffuser. The performance of the diffuser is dependent on the geometry and the downstream elements. To simulate the down-stream elements, small straight length is incorporated on the down stream of the diffuser with multiple annulus openings (Fig. 4) to simulate the actual flow resistance offered by the substrate\(^7\). Multiple annulus openings have been incorporated in the outlet pipe and the height of each annulus opening was adjusted to achieve the same pressure loss as determined experimentally in a conventional diffuser\(^6\).

Having modeled the back pressure conditions, the following geometries have been investigated (All dimensions given in the figure are in mm):

(i) Conventional diffuser — Fig 5a
(ii) Conventional diffuser with increased inlet diameter — Fig 5b
(iii) Enhanced Diffuser Header (EDH) — Fig 5c
(iv) Two stage diffuser (1) — Fig 5d
(v) Two stage diffuser (2) — Fig 5e
(vi) Two stage short diffuser (1) — Fig 5f
(vii) Two stage short diffuser (2) — Fig 5g
(viii) Two stage short diffuser (3) — Fig 5h

In these sets of the diffusers, diffusers (i), (ii) and (iii) are investigated to validate the CFD Code for similar geometries. Next two diffusers (iv) and (v) are...
KULKARNI et al.: OPTIMUM DIFFUSER GEOMETRY FOR THE AUTOMOTIVE CATALYTIC CONVERTER  

Fig. 5a—Geometry of conventional diffuser

Fig. 5b—Conventional diffuser with increased inlet pipe diameter

Fig. 5c—EDH (Enhance Diffuser Header)

Fig. 5d—2-Stage diffuser (N=155.75 mm, $\phi_2=30^\circ$)

Fig. 5e—2-Stage diffuser (N=155.75 mm, $\phi_2=15^\circ$)

Fig. 5f—2-Stage short diffuser (N=48.5 mm, $\phi_1=15^\circ$, $\phi_2=45^\circ$)

Fig. 5g—2-Stage short diffuser (N=48.5 mm, $\phi_1=20^\circ$, $\phi_2=45^\circ$)

Fig. 5h—2-Stage short curved diffuser (optimum geometry)
<table>
<thead>
<tr>
<th>Description</th>
<th>Normalised pressure loss ($\xi$)</th>
<th>Pressure recovery coefficient ($C_p$)</th>
<th>Diffuser effectiveness in % ($\eta$)</th>
<th>Maldistribution index at Distance ($X$, mm)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conventional diffuser (Fig. 5a)</td>
<td>0.25</td>
<td>0.70</td>
<td>73</td>
<td>$M_{x=0}=0.09$</td>
<td>1) High pressure loss due to flow separation</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=22.4}=0.51$</td>
<td>2) Flow considerably non-uniform at the substrate entrance</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=48.5}=0.96$</td>
<td></td>
</tr>
<tr>
<td>Conventional diffuser with increased inlet pipe diameter (Fig. 5b)</td>
<td>0.128</td>
<td>0.63</td>
<td>81</td>
<td>$M_{x=0}=0.09$</td>
<td>1) Overall pressure loss reduced by 48.8% compared to 0.46 mm inlet dia</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=13}=0.14$</td>
<td>2) Overall increase in diffuser effectiveness =17% because of reduced AR</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=27.7}=0.65$</td>
<td>3) Improved velocity profile since increased diameter of separated jet flow gases</td>
</tr>
<tr>
<td>Enhance diffuser header reference $^7$</td>
<td>0.14</td>
<td>0.816</td>
<td>85</td>
<td>$M_{x=0}=0.053$</td>
<td>1) Overall pressure loss reduced by 44% compared to conventional diffuser</td>
</tr>
<tr>
<td>$L=155.75$ mm (Fig. 5c)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=24.6}=0.062$</td>
<td>2) Overall increase in diffuser effectiveness =16% &amp; $M$ improved by 20%</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=155.75}=0.77$</td>
<td>3) Improvement in the parameters is at the cost of large length which is three times more than the conventional length</td>
</tr>
<tr>
<td>Two stage diffuser $^7$</td>
<td>0.13</td>
<td>0.82</td>
<td>86</td>
<td>$M_{x=0}=0.055$</td>
<td>1) Overall pressure loss reduced by 48% almost equal to the result achieved by increasing inlet pipe dia. in case of conventional diffuser</td>
</tr>
<tr>
<td>$L=155.75$ mm $\phi=30^\circ$ (Fig. 5d)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=16.5}=0.053$</td>
<td>2) Results are better than EDH geometry for the same length</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=155.75}=0.74$</td>
<td></td>
</tr>
<tr>
<td>Two stage diffuser $^7$</td>
<td>0.13</td>
<td>0.827</td>
<td>86.3</td>
<td>$M_{x=0}=0.05$</td>
<td>1) Slight improvement in pressure loss and diffuser effectiveness</td>
</tr>
<tr>
<td>$L=155.75$ mm $\phi=15^\circ$ (Fig. 5e)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=16.8}=0.17$</td>
<td>2) On other side flow becomes highly non uniform because of early flow separation</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=155.75}=1.03$</td>
<td></td>
</tr>
<tr>
<td>Through diffuser $^7$</td>
<td>0.25</td>
<td>0.7</td>
<td>73</td>
<td>$M_{x=0}=0.085$</td>
<td>1) For the same length through diffuser has given same pressure loss as that of in the conventional design</td>
</tr>
<tr>
<td>$L=155.75$ mm $\phi=10.19^\circ$ (Fig. 5e same length single stage)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=12.3}=0.17$</td>
<td>2) But $M$ is improved by 27% because of shallow and steady diffusion</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=155.75}=0.7$</td>
<td></td>
</tr>
<tr>
<td>Two stage short diffuser $^7$</td>
<td>0.2</td>
<td>0.74</td>
<td>77</td>
<td>$M_{x=0}=0.08$</td>
<td>1) Pressure loss improved by 20%</td>
</tr>
<tr>
<td>$L=48.5$ mm $\phi=15^\circ$ $\phi=45^\circ$ (Fig. 5f)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=23.4}=0.172$</td>
<td>2) Maldistribution index is increased by 14.5% because of sudden and steep diffusion and hence large scale separation in later part</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=48.5}=1.1$</td>
<td></td>
</tr>
<tr>
<td>Two stage short diffuser $^7$</td>
<td>0.08</td>
<td>0.85</td>
<td>89.6</td>
<td>$M_{x=0}=0.127$</td>
<td>1) Pressure loss reduced by 68% while diffuser effectiveness also increased by 22% which is best among above discussed geometries</td>
</tr>
<tr>
<td>$L=48.5$ mm $\phi=20^\circ$ $\phi=45^\circ$ (Fig. 5g)</td>
<td></td>
<td></td>
<td></td>
<td>$M_{x=24.7}=0.27$</td>
<td>2) But $M$ is also increased by 14.5% which is not acceptable</td>
</tr>
</tbody>
</table>
selected to study the effect of the geometry on the diffuser performance having geometrical dimension identical to the EDH geometry. The last three investigated diffusers [diffusers (vi), (vii) and (viii)] had shorter length but have same inlet and exit dimensions. These diffusers are investigated to study the effect of length, as this is a major constraint in the installation of a catalytic converter.

Results and Discussion
The results for all the investigated diffusers are tabulated in Table 1. For comparison, the following parameters have been evaluated:

(i) Normalized pressure loss
\[ \xi = \frac{P_{11} - P_{22}}{\frac{1}{2} \rho U_1^2} \]  \hspace{1cm} \text{... (19)}

(ii) Pressure recovery coefficient
\[ C_p = \frac{P_{32} - P_{31}}{\frac{1}{2} \rho U_1^2} \]  \hspace{1cm} \text{... (20)}

(iii) Diffuser effectiveness
\[ \varepsilon = \frac{C_p}{\text{Ideal Pressure Recovery}(C_{p\text{r}})} \]  \hspace{1cm} \text{... (21)}

where, \( C_p = 1 - 1/A_{R^2} \)

(iv) Maldistribution function
\[ M=\frac{(V_{\text{peak}}-V_{\text{avg}})}{V_{\text{avg}}} \]  \hspace{1cm} \text{... (22)}

From the results given in Table 1, it is seen that for conventional diffusers, the pressure loss is high but increasing the inlet diameter and reducing the diffuser half angle for same length, pressure loss reduces by 49% and the effectiveness increases by 11%.

For EDH, the pressure loss reduces by 44%, diffuser effectiveness increases by 16% and maldistribution index improves by 20% compared to conventional diffuser but at the expense of increased length. For improved EDH geometries, there is marginal improvement in pressure loss and diffuser effectiveness.

For modified conventional geometries designated as short diffusers, the pressure loss, diffuser effectiveness and maldistribution function have been compared with the conventional diffuser. It is seen that short diffuser (Fig 5g) gives the minimum pressure loss but the maldistribution index is not acceptable. The short diffuser geometry given in Fig 5h gives a slightly higher pressure loss but has an improved maldistribution index. To clearly bring out the reasons for improvement, the velocity distribution
in the conventional diffuser and the modified short diffuser are given in Figs 6a and 6b. It is seen that there is marked improvement in the velocity within the diffuser, which leads to the improved overall performance of the short diffuser.

The performance of the diffuser (Fig. 5h) identified as the optimum diffuser was investigated for increased inlet temperature and non-uniform inlet velocity distribution. The flow of hot gases enters the catalytic converter at higher temperatures and as the catalytic converter is placed downstream of 90° bend, the velocity profile at the inlet header of the converter is highly skewed with the peak velocity shifted from center. This velocity profile has been simulated and has been identified as non-uniform velocity profile. For increased inlet temperature from 20°C to 800°C, pressure loss increased by a factor of 8 whereas recovery coefficient and effectiveness decreased by only a factor of 4. For non-uniform velocity profile, there is a slight adverse effect on pressure loss, pressure recovery and diffuser effectiveness but significant effect is seen on maldistribution index (Table 1).

**Conclusions**

Although the flow in the catalytic converter is three-dimensional, two-dimensional axisymmetric computational analysis gives fairly accurate results and hence can be used for optimizing the design.

Gradual diffusion in number of stages improves pressure coefficient, diffuser effectiveness and maldistribution index.

 Provision of curvature at the interfaces of sudden changes of cross sectional area is useful as it delays the flow separation which otherwise takes place at the sharp edge itself. The optimum design of the diffuser is the one having an intermediate diffuser half angle for most part of its length and then gradual but steeper diffusion in the later part along a smooth wall curvature (Fig. 5h).

At higher temperatures, the optimum geometry (Fig. 5h) is observed to give better results than any other geometry at the same conditions. The non-uniformity in the velocity profile at the diffuser inlet increases the maldistribution index at the diffuser exit and normalized pressure loss increases by 0.035 while with same velocity profile it increased by 0.06 for conventional diffuser.

**Nomenclature**

- $A_R$ = Area ratio
- $C_{pr}$, $C_1$, $C_2$ = Turbulent model constants
- $C_p$ = Pressure recovery coefficient
- $G$ = Rate of production of turbulent kinetic energy
- $k$ = Turbulence kinetic energy, N-m
\[ M = \text{Maldistribution function} \]
\[ P_s = \text{Static pressure, N/m}^2 \]
\[ P_t = \text{Total or stagnation pressure, N/m}^2 \]
\[ \bar{S}_{ij} = \text{Source terms of velocity} \]
\[ q_0 = \text{Turbulent flux} \]
\[ \bar{T}, \bar{\tau}_{ij} = \text{Stress tensor, N/m}^2 \]
\[ \vec{U} = \text{Velocity Vector, m/sec} \]
\[ \bar{U}_i = \text{Average velocity at inlet, m/sec} \]
\[ u_t = \text{Fluctuating components of velocity, m/sec} \]
\[ V_{\text{avg}} = \text{Average velocity at any cross-section, m/sec} \]
\[ V_{\text{peak}} = \text{Maximum velocity at any cross-section, m/sec} \]
\[ x = \text{Cartesian co-ordinates} \]

**Greek letters**

\[ \alpha_0 = \text{Projection of the base vectors of the general co-ordinates system on to the cartesian base vectors} \]
\[ \frac{A}{\Delta x} = \text{differential operator involving the Jacobian of co-ordinates transformation} \]
\[ \varepsilon = \text{Turbulence Dissipation rate, N m}^2/\text{kg} \]
\[ \Phi = \text{Scalar variable (k, r, p .......)} \]
\[ \phi_1 = \text{Fluctuating components of velocity, m/sec} \]
\[ \mu = \text{Laminar viscosity, Ns/m}^2 \]
\[ \mu_t = \text{Turbulent viscosity, Ns/m}^2 \]
\[ \sigma_c, \sigma_g = \text{Constants in Turbulent Model} \]
\[ \Gamma_0 = \text{Turbulent diffusivity} \]
\[ \xi = \text{Normalized loss coefficient} \]

**Subscripts**

1 = Inlet condition
2 = Outlet or exit condition
i,j = 1, 2, 3

**References**