Numerical Simulation of Flow Field and Pollutant Dispersion in A Long Highway Tunnel with Semi-Transverse Ventilation

B.K. Gandhi, Jitendra Kumar and K.M. Singh

Abstract—This paper presents CFD analysis of flow field and pollutant dispersion in a long highway tunnel with semi-transverse ventilation in the middle Himalayas. A commercial CFD code (CFX) has been used for numerical simulation of the turbulent flow and scalar transport of carbon monoxide (CO) and nitrogen oxides (NOx). Vehicular emissions have been modelled as point sources. Standard k-ε turbulence model has been used in steady state flow analysis. Presence of fine sand in the air has been accounted for by assuming the working fluid to be a homogeneous mixture of sand and air. Simulation results provide an estimate of variation of the pollutant concentrations in different parts of the tunnel and clearly show the effect of variable percentage of fine dust/sand on ventilation in the tunnel.

Keywords— Pollutant dispersion; Semi-transverse ventilation; Tunnel ventilation; CFD.

I. INTRODUCTION

Tunnels form an important part of highway systems across the world in urban as well as mountainous regions. These can be of varying dimensions depending on the requirements and topography of the region. Vehicular emissions pose a serious threat to the quality of air in a tunnel, and provision of a proper ventilation system is mandatory to ensure an acceptable air quality. Design of the ventilation system for a particular tunnel depends on variety of factors such as the local terrain and length of the tunnel. Consequently, a tunnel may have natural ventilation (with or without provision of shafts for extraction of exhaust due to stack effect) or a forced ventilation systems consisting of an array of fans and ducts for supply/extraction of air [1].

With the significant progress in computing hardware and software in past few decades, numerical simulation has become an essential tool for analysis of engineering systems. Availability of commercial CFD codes with user-friendly graphical interface for geometric modelling and data analysis has provided further impetus to use of CFD simulation in analysis of a wide class of thermo-fluid problems. CFD simulations have been used for analyzing air quality (in terms of distribution of pollutant dispersion), effectiveness of ventilation strategies and for determining optimum location of mechanical devices (fans, ventilation vents and shafts) [2-7]. Numerical simulation has also been used to simulate spread of accidental fire and effectiveness of ventilation system for smoke extraction [8-10].

The present work is part of a sponsored investigation of an existing naturally ventilated double lane long highway tunnel in the middle Himalayas. It has two vertical shafts at equal distance from the portals for extraction of the polluted air by stack effect. Natural ventilation in the tunnel is aided by inflow of air from adits and exhaust from vertical shafts. Concerns have been raised about the effectiveness of the existing ventilation system, which have led to this investigation with two objectives: (a) analyze the effectiveness of the ventilation system, and (b) suggest suitable measures to improve the ventilation in the tunnel for the current maximum traffic rate. The tunnel design, traffic data and environmental conditions at the location of the tunnel have been provided by the sponsoring agency which runs the tunnel. Onsite measurements were done by the authors to collect data on velocity and pressure distribution, and finer geometric details which were not clear from the design drawings. Numerical simulation using a commercial CFD code is the primary tool for analysis. First part of this investigation focused on the the first objective of investigation of effectiveness of the ventilation system by analyzing the flow field and pollutant concentration obtained from numerical simulation [11-12]. This paper is concerned with the second objective, namely suggestion of suitable measures for improved ventilation to meet ventilation requirements for projected growth in highway traffic in the future. A semi-transverse ventilation systems has been suggested and numerical analysis of its effectiveness has been presented in this paper.

We start with a brief outline of the mathematical modeling in the next section. This is followed by the detailed description of the geometry modeling, mesh generation and numerical simulation process, results and concluding remarks in the succeeding sections.
II. MATHEMATICAL MODELING

The air inside the tunnel is taken as homogenous fluid for the flow field analysis and the concentrations of exhaust gases (i.e. CO and NOx) are taken as the additional scalar variables for the pollutant dispersion in the fluid domain. Simulation of the flow and pollutant dispersion in the tunnel involves numerical solution of the following conservation equations with appropriate set of boundary conditions:

- Continuity equation:

  \[
  \frac{\partial \rho}{\partial t} + \nabla . (\rho \mathbf{v}) = 0
  \]

- Momentum equation:

  \[
  \frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla . (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla . (\mu \nabla \mathbf{v}) + \mathbf{b}
  \]

- Energy equation:

  \[
  \frac{\partial (\rho i)}{\partial t} + \nabla . (\rho \mathbf{v} i) = -p \nabla . \mathbf{v} + \nabla . (k \nabla T) + \Phi + S_i
  \]

- Species transport equation:

  \[
  \frac{\partial (\rho \phi)}{\partial t} + \nabla . (\rho \phi \mathbf{v}) = \nabla . (\Gamma \nabla \phi) + S_\phi
  \]


The exhaust emissions for vehicles are modelled as point sources a fixed distance apart located 500 mm above the ground. The concentration of CO and NOx are taken according to Euro II standards norms. The standard k-ε model is used for the turbulence modelling with standard wall functions. The following assumptions are made for numerical simulation:

1. The flow is considered as steady state flow.
2. The mixture of sand particles and air is considered as a single phase fluid.
3. Exhaust gases do not affect the flow field in the tunnel.
4. The piston effect generated due the vehicles is neglected.
5. Tunnel wall is assumed a rough wall having sand grain roughness equivalent of 0.2 m.

Actual direction of airflow at the end portals, adits and shafts could be either into or out-of the tunnel. Hence, these have been modelled as opening boundaries. Walls of the tunnel, shafts and adits have been modelled as rough walls.

III. GEOMETRIC MODELLING AND NUMERICAL SOLUTION

A. Modeling of Tunnel Geometry

Figure 1 shows the schematic diagram of tunnel (including its cross-section, top view, ventilation shafts and adits). The tunnel consists of two circular ventilation exhaust shafts of 2.5 m diameter and 100 m high at reference distance RD 574 m and RD 1637 m respectively.

The exhaust emissions for vehicles are modelled as point sources a fixed distance apart located 500 mm above the ground. The concentration of CO and NOx are taken according to Euro II standards norms. The standard k-ε model is used for the turbulence modelling with standard wall functions. The following assumptions are made for numerical simulation:

1. The flow is considered as steady state flow.
2. The mixture of sand particles and air is considered as a single phase fluid.
3. Exhaust gases do not affect the flow field in the tunnel.
4. The piston effect generated due the vehicles is neglected.
5. Tunnel wall is assumed a rough wall having sand grain roughness equivalent of 0.2 m.

To improve the ventilation in the tunnel for anticipated traffic growth, we have proposed a semi-transverse ventilation system. It consists of a duct along the roof of
tunnel above the traffic lane having cross-sectional area of 4m×2m (Figure 1d) with 20 axial exhaust fan (each with a discharge capacity of 25 m³/s) mounted beneath it. Further, two fans of high discharge capacity (300 m³/s) are installed on the shafts to remove the concentration of contaminants from the duct through the shaft to avoid the choking.

B. Numerical Simulation Process

The set of governing equations (1-4) are solved subject to the boundary conditions by using CFX software. The physical domain has been discretized into a number of smaller, non overlapping sub domains in order to solve the flow physics [9]. A hybrid mesh is used for the discretization (Fig. 2). The first order upwind scheme is used for the advective terms. The convergence criterion for iterative solutions for all the equations has been taken as 10⁻⁵.

IV. RESULTS AND DISCUSSION

The tunnel under consideration is located in mountainous terrain adjacent to a river bed. Large amount of fine sand is brought inside the tunnel by vehicles passing through it. The actual concentration of sand in the air is highly variable depending on external weather conditions and seasonal variations which affect the amount of sand being brought into the tunnel by vehicle tyres and tractor trolleys. This variability is difficult to measure or predict accurately. Hence, to account for this variability, we performed a set of simulations with different volumetric concentration of sand in the air listed in Table I.

Pollutant dispersion in the tunnel has been analyzed at four different heights from the ground level namely at 1.5 m, 2.5 m, 3.5 m and 4.5 m which are represented by Line 1, Line 2, Line 3 and Line 4 respectively in further discussion. The total length of the tunnel is represented by 34 chart counts. Chart count 0 and 34 represent left and right ends of the tunnel (as shown in Figure 1).

Numerical simulations have been performed for all the seven cases listed in Table 1 for smooth as well as rough-wall conditions. We present representative results corresponding to two extreme cases (Case 1 and Case 7) in Figures 3-6.

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Sand Concentration (C_{0s})</th>
<th>Mean Density of Fluid (kg/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>1.2</td>
</tr>
<tr>
<td>2</td>
<td>0.00189</td>
<td>1.24</td>
</tr>
<tr>
<td>3</td>
<td>0.01</td>
<td>1.42</td>
</tr>
<tr>
<td>4</td>
<td>0.0189</td>
<td>1.63</td>
</tr>
<tr>
<td>5</td>
<td>0.039</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td>0.05</td>
<td>2.35</td>
</tr>
<tr>
<td>7</td>
<td>0.1</td>
<td>3.5</td>
</tr>
</tbody>
</table>

These figures show the distribution of pollutant concentration at different heights from the ground along the length of the tunnel. In all these plots, top curve corresponds to Line 1, second curve from top to Line 2, third curve from top to Line 3, and bottom curve corresponds to Line 4. All these plots show similar trends. Concentrations levels are higher in the left part of the tunnel and all pollutant concentrations decrease downstream for a fixed ventilation condition due to dilution of fresh air intake for adits and exhaust from the ventilation shafts. The plots
also reveal that pollutant concentration decreases along vertical direction in the tunnel and higher concentrations are present near the tunnel floor.

Figure 4: NOx concentration in the tunnel for Case 1

(a) Tunnel with natural ventilation
(b) Tunnel with semi-transverse ventilation

Figure 5: CO concentration in the tunnel for Case 7

(a) Tunnel with natural ventilation
(b) Tunnel with semi-transverse ventilation
From Figures 3-6, we can clearly observe the decrease in concentration of pollutants (CO and NOx) by a factor of half in all the situations. This observation confirms the effectiveness of the proposed transverse ventilation system for projected future increase in the traffic (and thus, increased pollutant load).

V. CONCLUDING REMARKS

We have presented a CFD analysis of the flow and dispersion of pollutants CO and NOx in a long highway tunnel in middle Himalayas. Vehicle emissions are modelled as point sources uniformly distributed at a height of 500 mm along the road. Source strength was based on the given peak traffic data and Euro II vehicle emission standards. Roughly cut tunnel walls have been modelled as a rough wall with a wall roughness of 0.2 m. Significant amount of fine sand present in the air has been accounted in the simulation by assuming the fluid in the tunnel to be a homogeneous mixture of air and sand. Numerical simulations clearly show the adverse effect of wall roughness on flow velocities and pollutant dispersion in the tunnel. Further, the volumetric concentration of sand has a significant effect on the flow field and pollutant dispersion in the tunnel. With increase in sand concentration, ventilation capacity is adversely affected. However, numerical simulations indicate that the maximum concentrations are well within the permissible limits. Further, the proposed semi-transverse ventilation system can lead to significant improvement in ventilation in the tunnel.

ACKNOWLEDGMENT

The authors gratefully acknowledge the financial support provided by HP State Electricity Board (Grant number MID-1013/10-11) for this work.

REFERENCES