Flow Behavior in a Rectangular Tunnel Opened and Closed at Both Sides Using CFD

Yong-Ho Lee†

(Received April 4, 2012; Revised April 20, 2012; Accepted April 20, 2012)

Abstract : Most tunnel simulations have been focused on the thermal field and the critical velocity for suppression of hot back-layering flow in case of fire and on the characteristics of a tunnel fire in terms of the flame propagation and the toxic gas generation. In this paper, a comparative study of the flow characteristics of polluted air with no heat source in a tunnel model opened and closed at both end sides is implemented into a recognized CFD simulation code. The model is used to investigate the flow characteristics depending on the three different Reynolds numbers of 640, 1270 and 2120, which have been chosen by the flow velocities of 0.3, 0.6 and 1.0 m/s through the inlet. The results of this study have shown that the CFD predictive and experimental approaches are available in qualitatively studying the correlation of flow behaviors for a better tunnel design.

Key words : Flow characteristics, Polluted air, Rectangular tunnel, PIV (Particle Image Velocimetry), CFD(Computational Fluid Dynamics)

1. Introduction

Quite a lot of types of similar rectangular tunnels are come into contact in our daily life. In a transportation system and a general ventilation system, those tunnels are used everywhere all the times. Thus, a special interest in safety regarding polluted air has grown significantly due to the increasing number of tunnel structures. Many investigations have been carried out focusing on smoke control for fire safety in those various tunnels. Basically, tunnels can be classified into three different types, which are closed at both end sides like a long corridor, closed at one end side like a coal mine tunnel, and opened at both end sides like a general road tunnel. The air flow in vertical or horizontal duct systems, in road or railroad tunnels, in highway tunnels, in subway tunnels, etc. is related to the pollution concentration due to the exhaust gas from vehicles or other accidents like poison gas attacks in Japanese subways in 1995. It is required to be designed for ventilation systems along the tunnel to control the polluted air in tunnels. Extreme damages caused by the polluted air in long corridors or tunnels have threatened the people's lives. In case of coal mine, the tunnel end is closed, whereas the tunnel outlet is open. Li Kun et al.[1] investigated the influence of ventilation tube rupture in a mine tunnel.

Barbason and Reiter[2] compared various turbulence models and reported that the standard k-ε model was a good compromise for various engineering applications except for natural ventilation with significant indoor thermal loads. Certain characteristics of wind flow with very low...
velocities and thus low Reynolds numbers could not be accurately predicted by standard k-ε model. A basic characteristic of the RNG k-ε turbulence model[3] is that it involves an analytically derived differential formula for effective viscosity that accounts for low-Reynolds number effects. Gebremedhin and Wu[4] stated that the RNG k-ε model was found to be the most appropriate model to characterize the flow field in a ventilated space based on convergence and computational stability criteria. Nguyen and Reiter [5] evaluated the effects of a building parameter, namely ceiling configuration on wind induced air motion inside a building by using the RNG k-ε model.

Hwang and Edwards [6] carried out a study on the critical ventilation velocity in inclined tunnels. Wu & Baker [7] investigated the values of the air velocities required to control the smoke movement in horizontal tunnels. The simulation results from smoke movement in longitudinally ventilated tunnels compared with experimental data were discussed [7,8]. Nuri et al. [9] conducted an experimental and numerical study to investigate fire behaviors in a scaled version of an underground station. The general flow pattern and the back-layering in velocity measured experimentally in the station at different locations were shown to be well matched by the CFD simulations.

The flow characteristics in a rectangular tunnel by providing instantaneous velocity vectors, mean vorticity fields, and instantaneous kinetic energy using a PIV system were discussed [10,11]. The nature of polluted air flow in a rectangular tunnel was presented by a comparative study using the PIV data acquisition and the CFD analysis [12]. The PIV system is possible to obtain instantaneous velocity maps, instantaneous vorticity maps, and instantaneous kinetic energy distributions in a plane flow field of interest due to the reliable field measurement technique and lots of special implementations [13,14].

If the fire source is considered and the flow velocity is low as well, hot smoke progressively spreads along the ceiling against the ventilation flow through the buoyancy forces generated by the fire. The critical velocity, required to suppress the backing-layer flow upstream of the fire section in the tunnel, has been studied [7,15].

In this study, however, the air flow with no heat source entrained through the inlet is considered to present the numerical results of air flow characteristics in two different type of rectangular tunnels at three different Reynolds numbers. The results have been obtained from the CFD using ANSYS CFX code. The air flow characteristics in two types of tunnels in connection with flow velocity and pressure distributions and laminar or turbulence kinetic energy are analyzed in order to provide some desirable information for the better tunnel design to be taken into consideration.

2. Physical Model for CFD Simulation

The rectangular tunnel model, as shown in Figure 1, is established based on the duct and ventilation systems or the road tunnel model. The tunnel model is 800 mm long and the dimension of the rectangular cross section is 80 mm × 80 mm. The mathematical model is set on the assumption that the air flow through the inlet is steady and the tunnel wall is supposed to be adiabatic. The RNG k-ε model is used to simulate the laminar or turbulent flow process. The inlet is located at the middle of the tunnel floor in the x-z plane, while the outlet is located on the ceiling of the tunnel model at L = 60 mm, in which L is defined as the distance from the vertical centerline of the inlet to the outlet. The dimension of both the inlet and outlet is 20mm × 80mm.
The air flow characteristics in the two different tunnel models have been researched at three different Reynolds numbers. The distance between the inlet and outlet are constant and three different Reynolds numbers due to the three different inlet velocities are given in this study. The velocity vector fields and streamlines, the pressure distributions, and the laminar or turbulence kinetic energy are discussed here.

3.1 The velocity vector fields

In order to show the flow velocity fields between the inlet and outlet, a cross-sectional image dimension of 140mm × 80mm has been examined by using CFD.

Figure 2 shows the velocity vector fields and streamlines in the two different tunnel models opened/closed at both end sides at the Reynolds number of 640. As shown in Figure 2(a), the small vortex is produced on the right upper part next to the outlet near the ceiling. The flow behavior at
very low Reynolds number directly leads to the tunnel ceiling regardless of the outlet geometry. However, in Figure 2(b) the flow goes up towards the outlet due to the pressure rise inside the tunnel closed at both end sides. The flow motion slopes up toward the outlet located at 60 mm away from the inlet centerline on the tunnel ceiling in the $x$ direction. It is also observed that the maximum value of velocity occurs in the outlet region in Figure 2(b).

**Figure 2:** Velocity vector fields and streamlines using CFD at $Re=640$

**Figure 3:** Velocity vector fields and streamlines using CFD at $Re=1270$
In Figures 3 and 4, both large recirculation regions are dividedly formed by the flowing zone passed through from the inlet to the outlet inside the tunnel model. The small vortex formed at the right upper part next to the outlet at Re=640 disappears as the Reynolds number is increased. The increase of inlet velocity results in failure to have a clockwise vortex next to the outlet on the right, but it grows a large recirculation region around the center of the tunnel in the y direction for both the tunnel opened/closed cases. This implies that some air flowing along the ceiling is discharged into the outlet, but the rest of air flows into the tunnel again.

Table 1: Inlet and outlet velocities in the tunnel opened/closed at both end sides. (m/s)

<table>
<thead>
<tr>
<th>Inlet velocity (Reynolds no.)</th>
<th>0.3 (640)</th>
<th>0.6 (1270)</th>
<th>1.0 (2120)</th>
</tr>
</thead>
<tbody>
<tr>
<td>outlet velocity</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>opened</td>
<td>0.316</td>
<td>0.481</td>
<td>0.695</td>
</tr>
<tr>
<td>closed</td>
<td>0.450</td>
<td>0.907</td>
<td>1.530</td>
</tr>
</tbody>
</table>

From the table 1, for the case of the tunnel opened at both end sides, the outlet flow rate at the lowest Reynolds number of 640 is equivalent of 105 percent of the inlet flow rate. However, as the Reynolds number is increased to 1270 and 2120, there is a rapid decrease of each outlet flow rate of 80 and 69 percents, compared to the each inlet flow rate. In the case of the tunnel opened at sides, this is the reason that the pressure inside the tunnel keeps the atmospheric pressure in all regions, even though there is a very little pressure difference near the inlet and outlet regions. Due to low pressure rise in the tunnel model, most half of the inlet flow rate is divided into two regions in both x directions at the top of the tunnel. Exceptionally at very low Reynolds number of 640, the outlet flow rate is higher than that through the inlet though the tunnel is opened at both sides.

For the case of tunnel closed at both end sides, the outlet flow rate in three cases reaches up from 150 to 153 percents, when compared to each inlet flow rate as the Reynolds number is increased. Without any special exception in the case of the tunnel closed at both sides, the outlet flow rates are higher than the inlet and the flow behavior appears
to be nearly identical. For the Re=1270 and R=2120 cases, the trends of air behavior in the tunnel are almost similar. This is considered to be a typical phenomenon caused by the effects of the pressure drop inside the tunnel closed at sides, even though the length of tunnel is longitudinally very long. It is available to make a decision to estimate quantitative information on concentrations and emission rates of pollutant gas.

3.2 Pressure Distributions Using CFD

Figures 5 to 7 show the pressure distributions in terms of two tunnel models as the Reynolds number increases. The contours for the pressure distribution in Figures 5(a), 6(a) and 7(a) are brought from the tunnel models opened at both end sides. The second cases in Figures 5(b), 6(b), and 7(b) are presented as the models closed at both end sides.

Figure 5: Pressure distributions using CFD at Re=640

(a) tunnel model opened at both sides

(b) tunnel model closed at both sides

Figure 6: Pressure distributions using CFD at Re=1270

(a) tunnel model opened at both sides

(b) tunnel model closed at both sides

Figure 7: Pressure distributions using CFD at Re=2120

(a) tunnel model opened at both sides

(b) tunnel model closed at both sides
As shown in Figure 5(a), the inlet pressure is slightly lower than the atmospheric pressure. The maximum pressure in the tunnel opened case here is closely at the ceiling just above the inlet position, while the lowest pressure near the ceiling is found at the outlet. In the tunnel closed case in Figure 5(b), the maximum pressure is moved close to the outlet and is positioned at the left side of the outlet. There is also a small region of a little high pressure next to the outlet on the right.

Figure 6 shows the pressure distributions at the Reynolds number of 1270. In the tunnel opened case in Figure 6(a), the ceiling region just above the inlet position reaches the highest pressure like the same as its previous case in Figure 5(a). The pressure distributions in other regions throughout the tunnel are almost similar. In Figure 6(b), higher pressure region is put on the left side of the border produced by the flow motion from the inlet to outlet, whereas lower pressure region is on the right side. The overall pressure in the tunnel closed case at Re=1270 is applied higher than that at Re=640.

In the case of the tunnel opened at both end sides in Figure 7(a), the position of maximum pressure is identical to the two previous tunnel opened cases and the pressure discharged through the outlet is lowest around the tunnel ceiling. On the other hand, in the case of the tunnel closed in Figure 7(b), the trend of flow field is, in effect, identical to that of the two previous tunnel closed cases as well.

Table 2: Inlet and outlet pressures in the tunnel opened/closed at both end sides. (Pa)

<table>
<thead>
<tr>
<th>Reynolds No.</th>
<th>640</th>
<th>1270</th>
<th>2120</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure _opened (Pa)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>inlet</td>
<td>-0.0294</td>
<td>-0.0785</td>
<td>-0.2001</td>
</tr>
<tr>
<td>outlet</td>
<td>0.00301</td>
<td>0.00062</td>
<td>0.00147</td>
</tr>
<tr>
<td>Pressure _closed (Pa)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>inlet</td>
<td>0.02101</td>
<td>0.29702</td>
<td>0.92906</td>
</tr>
<tr>
<td>outlet</td>
<td>0.00307</td>
<td>0.00928</td>
<td>0.90883</td>
</tr>
</tbody>
</table>

In the case of the tunnel opened at both end sides, the pressure difference between outlet and inlet is defined as the pressure rise, \( \Delta P_{\text{opened}} = P_{\text{outlet}} - P_{\text{inlet}} \). The pressure rise in the tunnel opened case increases as the Reynolds number is increased, but the pressure rise of \( \Delta P_{\text{opened}} \) is very small because the tunnel is open at both sides to the atmospheric pressure. However, in the tunnel closed at both end sides, the pressure drops occur because the inlet pressure is higher than the outlet. The pressure drops, \( \Delta P_{\text{closed}} = P_{\text{inlet}} - P_{\text{outlet}} \), are caused by the pressure difference between the inlet and outlet inside the tunnel. Thus, the pressure in the tunnel opened case rises, whereas the pressure in the tunnel closed case drops at each different Reynolds number. From table 2, the ratio of pressure rise in the tunnel opened case at very low Reynolds number of 640 is 1.7 times higher than that in the tunnel closed case. On the other hand, as the Reynolds number increases, the ratios of pressure drop in the tunnel closed case are 3.6 and 4.5 times higher than those in the tunnel opened case. The variation of pressure through the inlet and outlet in the tunnel opened and closed cases can be investigated by the CFD analysis.

3.3 Kinetic Energy Field

The kinetic energy of an objects is the energy which it possesses due to its motion[16]. It is defined as the work needed to accelerate a body of a given mass from rest to its stated velocity.

In Figure 8(a), which shows the laminar kinetic energy at very low Reynolds number of 640, most kinetic energy is distributed along both sides of the path which is moving along the flow direction from the inlet to outlet. So the maximum energy is measured near the outlet with the bigger variations in velocity during any small time interval. In Figures 9(a) and 10(a), as the Reynolds number
increases, the effect of the outlet on the ceiling is reduced due to the exposure to atmospheric pressure in the tunnel opened at both end sides. In this case the maximum energy is concentrated at the ceiling just above the inlet position.

For the tunnel closed at both end sides, the kinetic energy fields are shown in Figures 8(b), 9(b) and 10(b). In these cases, the trend of kinetic energy distributions is almost similar in following the direction of flow motion, although the amounts of laminar or turbulence kinetic energy are a slight difference among the three tunnels due to the different Reynolds numbers. For all cases in the tunnel closed at both end sides, the kinetic energy is distributed, with tilted slightly to the right, along both sides of the path from the inlet to the outlet. The flow behavior is most active around the outlet region regardless of the range of the Reynolds number, but the amount of kinetic energy based on the motion of the polluted air has a considerable difference depending on the Reynolds number.

5. Conclusions

Good agreement between the CFD simulations and the validation experiments is not easy to achieve. However, a CFD analysis is desirable to
better understand the nature of polluted air through the use of computational models before the experiment in advance. Here are two CFD models to predict the flow characteristics. One is the normal tunnel opened at both end sides and the other is the tunnel closed at both end sides like a long corridor. The current paper investigated the velocity vectors and streamlines, the pressure distributions, and the laminar/turbulence kinetic energy contours by producing the predicted results using CFD.

(1) For the tunnel opened at both end sides, inlet velocity does not exert a strong influence on the change of flow rate in the outlet although the Reynolds number increases. However, at very low Reynolds number, the outlet flow rate is a little more positive than the inlet. It is considered that low laminar flow may not be strongly affected by the flow velocity.

(2) For the tunnel closed at both end sides, the outlet flow rate is about 1.5 times higher than the outlet regardless of the change of the Reynolds numbers.

(3) In the case of the tunnel opened at both end sides, the inlet pressure is a very little lower than the outlet pressure and the pressure difference between the inlet and outlet is smaller than that in the tunnel closed case. In the case of the tunnel closed at both end sides, the pressure drop is caused because the inlet pressure is higher than the outlet, while the pressure rise is produced in the tunnel opened case. In particular, at very low Reynolds number the pressure difference between the inlet and outlet is very slight, when compared to other Reynolds numbers.

(4) The kinetic energy is originated from the flow motion inside the tunnel due to the flow velocity. At very low Reynolds number in the tunnel opened case, the maximum kinetic energy is shown in the region of vortex shedding next to the outlet on the right side of the ceiling. Similar trends in each energy distribution along both sides of the path followed by the direction of flow motion are seen for all cases, although the amounts of laminar or turbulence kinetic energy are a considerable difference due to the different Reynolds numbers.

References


Author Profile

Yong-Ho Lee
He obtained his B.S. in Mechanical Engineering from Chosun University in 1983. He got his M.S. from Inha University in 1985, his Ph.D. from Chonnam National University in 1999, and his Postdoctoral Certificate from Tsinghua University, China, in 2002. He is an instructor of the School of Mechanical and Automotive Engineering at Chonnam National University, Yeosu Campus.