5.10 MODELLING POLLUTANT DISPERSAL AT THE PORTALS OF ROAD TUNNELS

Gourdol, F.[1], Perkins, R.J.[2], Carlotti, P.[1], Soulhac, L.[2] & Méjean, P.[2]
[1]Centre d'Etudes des Tunnels, France
[2]Laboratoire de Mécanique des Fluides et d'Acoustique, CNRS UMR 5509
Ecole Centrale de Lyon, France

INTRODUCTION

The pollution emitted by vehicles inside a road tunnel is often vented directly to the atmosphere at the tunnel portals. Because the tunnel is an enclosed space, the pollutant concentrations are higher than in the open air, and in an urban environment the concentrations at the portals can exceed regulatory limits. The likely impact of tunnel emissions on urban air quality can be estimated using techniques such as 3-D numerical simulations, or physical modelling of a specific tunnel in a wind tunnel or a hydraulic flume, but all these methods suffer from several important problems. Firstly, they are relatively costly to implement (both in time and in money) and are difficult to use in impact assessments where it is necessary to study a large number of different conditions. Secondly, they all require a number of simplifying assumptions, concerning both the phenomena to be modelled and the boundary conditions to be used, and the results are often very sensitive to the assumptions that are made.

We have therefore studied dispersion in the neighbourhood of a tunnel portal, with the aim of identifying the basic phenomena concerned, and compiling a database. The second step is to develop a simple model that can be used in impact studies.

EXPERIMENTAL SETUP AND MEASUREMENT TECHNIQUES

The experiments were performed in the atmospheric boundary layer wind tunnel of the Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), France. The road tunnel was modelled as a simple partially-covered cutting, set into the floor of the wind tunnel (Figure 1). The tunnel orientation could be varied relative to the wind direction, and experiments were conducted for 7 different wind directions (0° to 180° in steps of 30°). The mean air velocity in the tunnel (U_r) could be adjusted independent of the wind speed (V) and experiments were performed for four different values of the wind speed ratio (U_r/V=0.25; 0.5; 1 and 2).

The boundary layer was generated using a combination of three spires located at the entrance

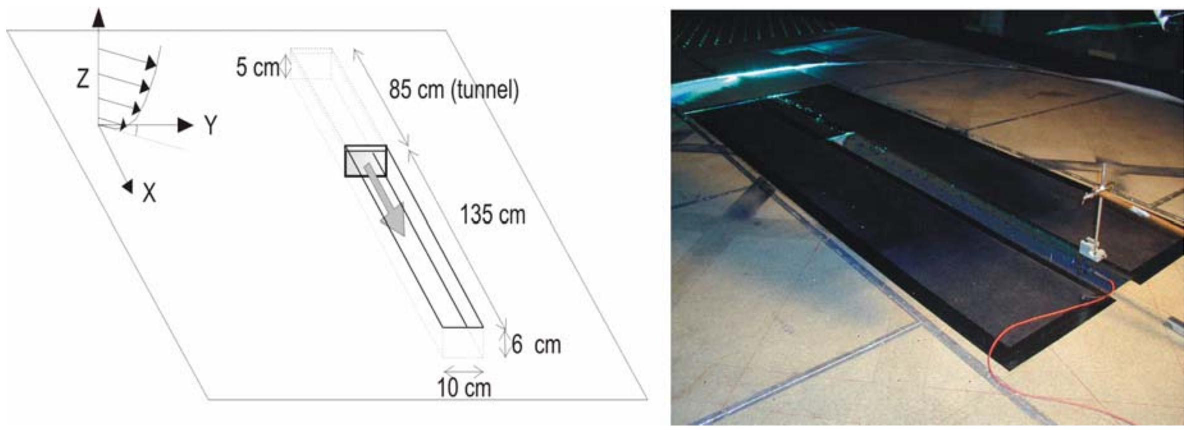

Figure 1. Small-scale model

to the test section, with a lateral spacing equal to half the spire height (Irwin, 1981) and small roughness cubes (height 2cm) on the floor of the tunnel. The velocity field close to the surface is well described by the classical logarithmic law profile with a roughness height of 0.14mm:

$$U(z) = \frac{u_*}{\kappa} \ln\left(\frac{z}{z_0}\right) \qquad (1)$$

Mean and fluctuating velocities were measured using a three-component laser Doppler velocimeter. Mean and fluctuating concentrations were measured using a Flame Ionisation Detector, with Ethane as the tracer gas. The Ethane was injected into the airflow in the tunnel, well upstream of the portal, so that the gas was well-mixed when it emerged into the flow. Velocities and concentrations were measured within the open part of the cutting, at various distances from the portal, and also at various locations downwind of the tunnel and cutting.

RESULTS

In operational studies it is usually assumed that the flow can be characterised by the ratio U_r/V, but as far as we are aware, this has never been shown to be true. We have therefore carried out a preliminary study to verify that this ratio is an appropriate parameter. A second set of experiments has been performed to study the influence of the wind direction relative to the tunnel axis.

1. The ratio U_r/V

In order to verify that the dimensionless velocity field U/U_0 depends only on the ratio U_r/V and not on the absolute velocity we have carried out a series of measurements for one wind direction (θ=90°) and constant velocity ratio, but different wind speeds in the range 1-4 m/s. The resulting velocity profiles (Figure 2) show that the main phenomena (recirculation flow in the street and an axial decrease in the jet velocity) are effectively independent of the imposed wind speed.

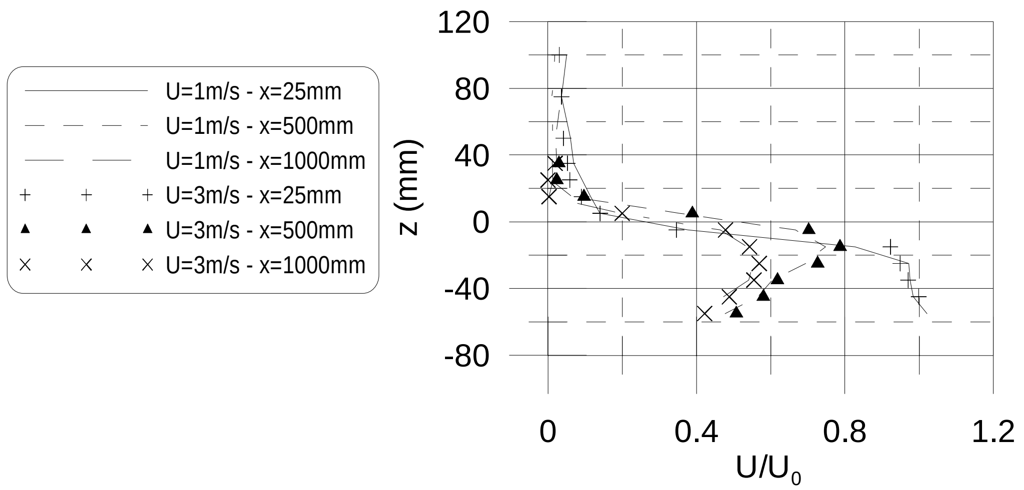

Figure 2. Velocity profiles showing independence of the ratio U_r/V

2. The influence of wind direction

θ=0°

The wind is blowing parallel to the tunnel axis, and in the same direction as the flow in the tunnel, so the pollutants remain trapped in the street canyon where high levels of pollution are found far from the tunnel exit. The lateral expansion of the plume is very limited, and there is little diffusion to the exterior of the cavity.

30° ≤ θ ≤ 90°

Within this range the dimensionless velocities and concentrations do not vary much with wind direction, so we have plotted here (figure 3) velocity and concentration profiles for an angle of incidence of 90°. The flow in the street canyon can be decomposed into two main zones, defined by the downwind distance from the tunnel exit. The first zone (near the exit) is characterized by the presence of strong cross-street concentration gradients, with highly contaminated air on the upwind side of the street. The axial velocity in the cutting U shows a similar behaviour, whilst the other two components, (V and W) vary with x (and of course with y and z).

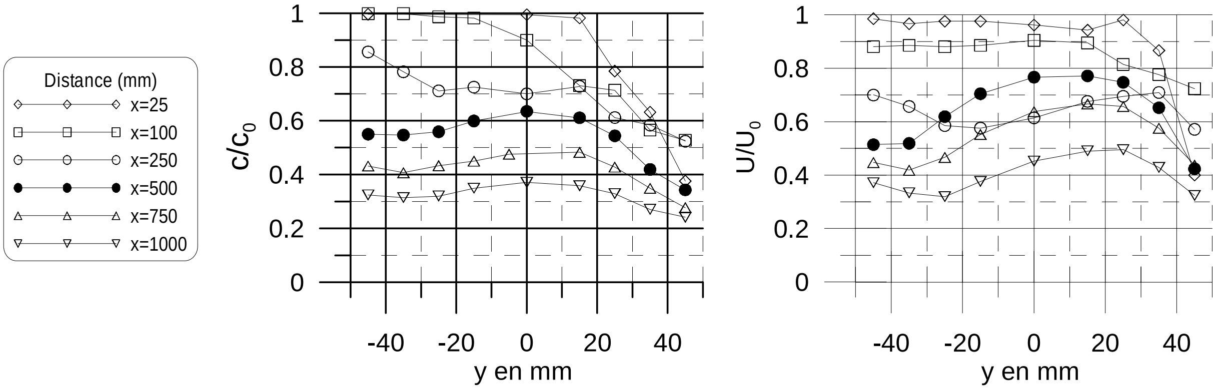

Figure 3. Evolution of the flow in the street canyon according to the distance at the exit – Profile at half- height of the street for 90° and a ratio of 0.5.

Further downstream from the tunnel exit the concentration and the axial velocity U both become much more homogeneous across the section homogenisation across the section. The other two velocity components become independent of x and the projection of the velocity field in the section is the one obtained for a street canyon without any axial flow along the canyon. The velocity field in the canyon can therefore be considered as a superposition of a recirculation flow (V(y,z) and W(y,z)) whose properties depend essentially on the wind perpendicular to the street as for the case of a street canyon (Soulhac, 2000), and an axial component U(x) which is here determined by the imposed flow in the tunnel.

Downwind of the street, the concentration field observed (Fig.4) at the ground level shows that the flux of pollutant out of the street is a maximum at some distance x_{max} downstream of the tunnel exit, where x_{max} varies with the speed ratio and the wind orientation.

120°≤θ≤180°

For these orientations, no recirculation in the street is observed. The main phenomenon is the reversal of the jet. Because of the reversal, the vertical spread of the jet is more important. As for the previous cases, the jet stays confined in the street and the wind is dominating outside the street.

CONCLUSION

The geometrical configuration chosen, a cut-and-cover follow by a street canyon, has a strong influence on the interaction between the ABL and the jet which is confined to within the street; this jet has little influence on the velocity field above the street, inducing mainly a deviation in wind direction. The knowledge of the pollutant flux at the interface is sufficient to estimate the concentration field in the external flow. This suggests that it ought to be possible to model the external concentration field by considering advection and dispersion from a line source of pollutant placed on the axis of the street. The problem is then to link the properties of this line source to parameters such as the speed ratio and the wind direction.

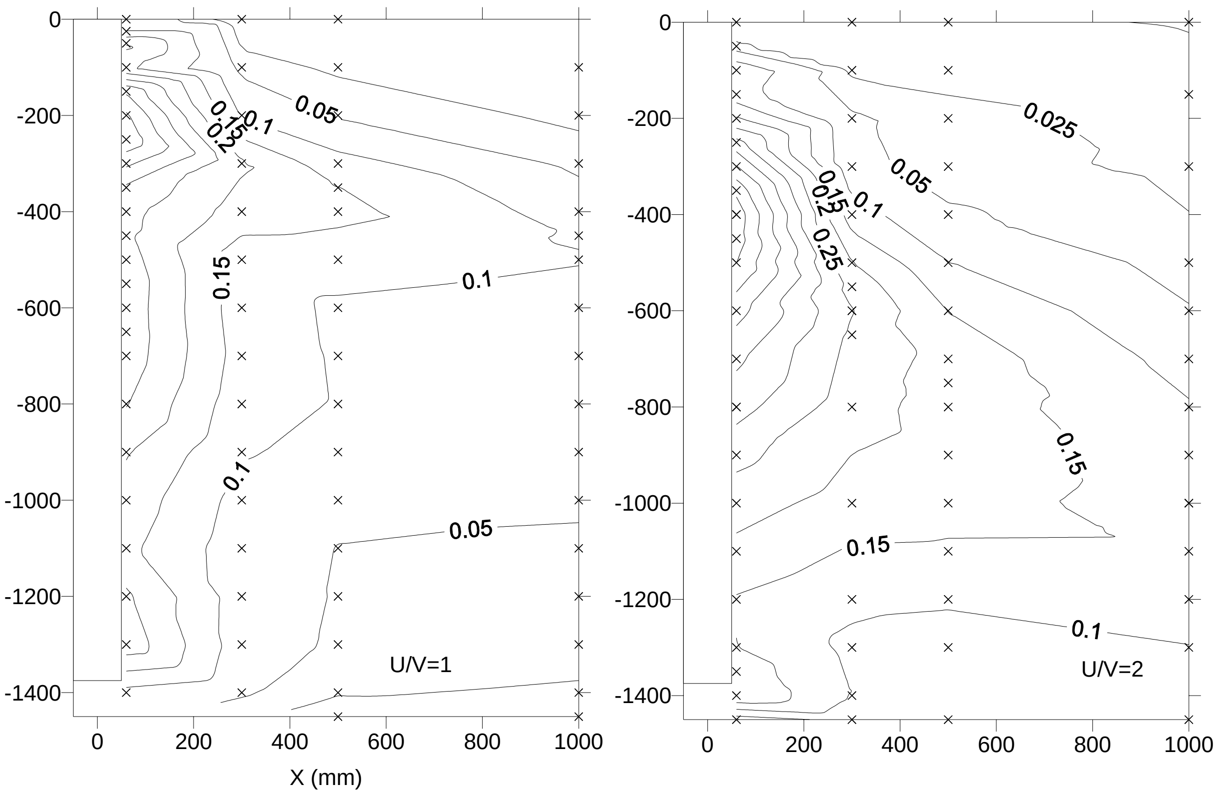

Figure 4. Concentration field at the ground downwind of the street – 90°

SIMPLE MODEL USING A LINE SOURCE

We propose to model the emission of pollutants from the street using a line source of pollutant placed along the axis of the street. The resulting concentration field can then be calculated using standard Gaussian plume type solutions. The major problem is then to determine how the strength of the line source should vary with distance downstream from the tunnel exit. Here we propose to determine the characteristics of the line source by comparing he calculated field with the measured field, and adjusting the coefficients of our line source strength distribution to maximise the agreement between predicted and measured concentration fields. An analytical law (with only two parameters) has been chosen to represent the line source emission in function of the distance x from the tunnel exit. As shown in Fig.5, such a model with suitable parameters can give results that agree reasonably well with measured data. But there are cases in which the model is inadequate, notably:

For strong jets (speed ratio of 2), it is necessary to take into account the deviation of the external flow by the jet in the cavity.

When jet reversal occurs, it is inappropriate to place the source on the axis of the street because of the enhanced vertical dispersion.

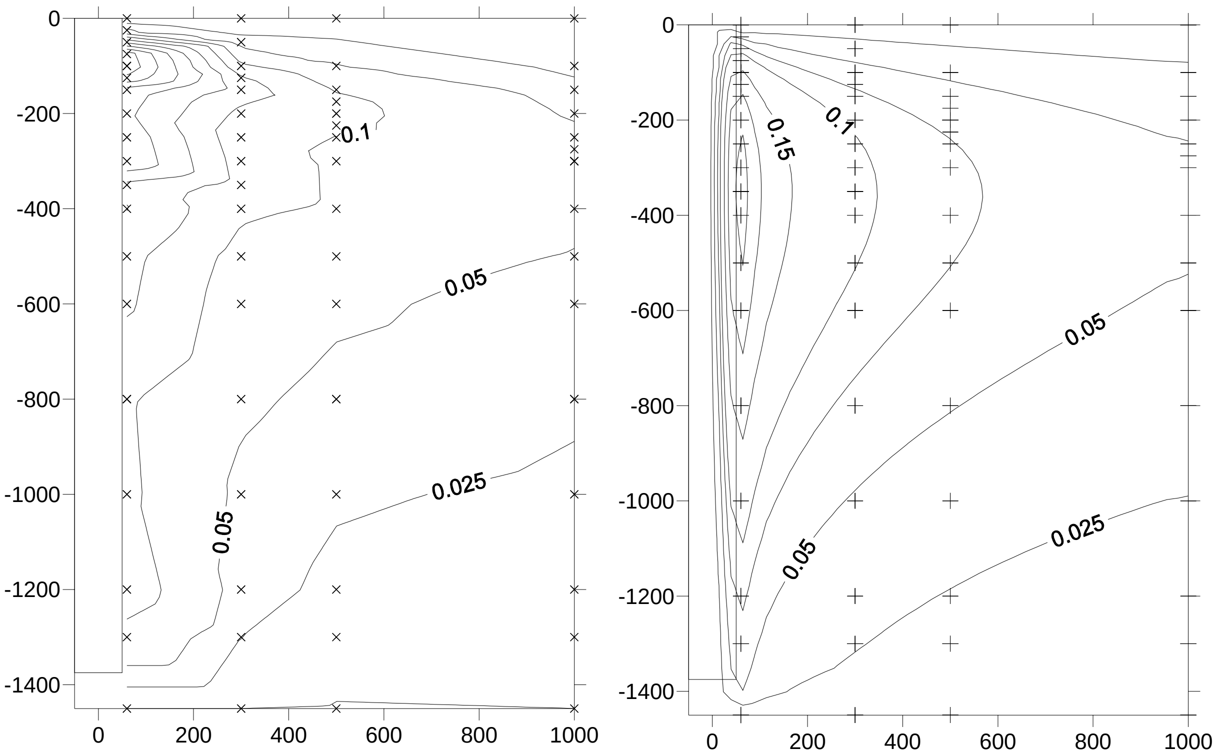

Figure 5. Comparison between measurements and simulation- 90° - Ur/V=0.5

CONCLUSION

An experimental database has been created for an idealised case. A general topology of the flow can be determined as a function of the wind orientation. Results have shown a preponderant influence of the geometrical configuration. Another experimental campaign is planned with another configuration: a cut-and-cover followed by a ramp.

The line source approach should be deeper investigated to take account of some phenomena: deviation of the plume due to the jet, enhanced vertical dispersion …

REFERENCES

Irwin, H.P.A.H., 1981: The design of spires for wind simulation. J. of Wind Eng. and Ind. Aero., **vol. 7**, pp 361-366.

Soulhac, L., 2000: Modélisation de la dispersion atmosphérique à l'intérieur de la canopée urbaine. PhD Thesis. Ecole Centrale de Lyon.