Dynamic Layering Method 를 이용한 터널 내 열차풍에 대한 수치해석
Numerical Study of Train-induced Unsteady Airflow in a Subway Tunnel Using Dynamic Layering Method

Y. D. Huang[1, 2], *홍태협 [2], #김창녕 [2]
Y. D. Huang[1, 2], *T. H. Hong[2] , #C. N. Kim[2] (cnkim@khu.ac.kr)
[1] University of Shanghai for Science and Technology, [2] 경희대학교 기계공학과

Key words : Train-induced flow, Numerical simulation, Tunnel, Moving boundaries, Dynamic layering method

1. Introduction

CFD simulation of train-induced unsteady flow in a subway tunnel is essential for optimizing the design of ventilation systems, and one of the main difficulties in developing the numerical method is how to model the moving boundaries of a train. There have been two methods developed to model the moving boundaries of an immersed solid: the moving grid method and the fixed grid method.

The application of moving grid method is closely related to the formation of a grid system. For the analysis of a train moving in a tunnel, the chimera grid system, the patched grid system, and the adaptive remeshing grid system were used [1]. The fixed grid method can be divided into the diffuse interface method and the sharp interface method [2]. Using the sharp interface method, Kim and Kim [1] have recently carried out a numerical analysis on the train-induced flow inside their experimental model tunnel.

In the current study, a numerical simulation on a train-induced tunnel flow is performed using the dynamic layering method based on the geometry and specific conditions obtained from the experimental study carried out by Kim and Kim [1] in a model tunnel. The calculated results are compared with the experimental data to evaluate the accuracy and applicability of the dynamic layering method.

2. Problem Formulation

2.1Experimental and numerical analyses by Kim and Kim [1]

Kim and Kim [1] conducted an experimental study on train-induced airflow in a model tunnel. Fig.1 shows a schematic diagram of the experimental layout. The tunnel is 39 m long, 250 mm high and 210 mm wide. The train is 3 m long, 225 mm high and 156 mm wide. The distance of the train run is 33 m. Two velocity transducers were installed 0.5 m inside the tunnel inlet and outlet，and four pressure transducers were installed 8.5 m, 15.5 m, 23.5 m, and 30.5 m from the tunnel inlet along the center of the model tunnel roof. The velocity of the experimental train run is shown in Fig.2.

Employing the geometric configurations and train-run velocity used in the experimental test, Kim and Kim [1] also conducted a numerical analysis on the train-induced unsteady flow inside the model tunnel. In their numerical simulation, the computational models were constructed and solved using the CFD software CFX4, and the sharp interface method was adopted to represent the moving boundaries of the train.

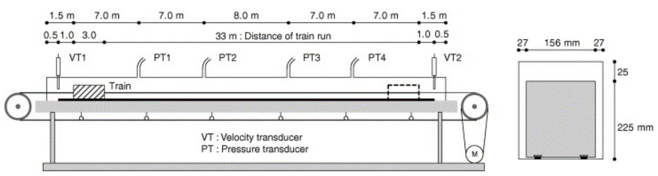

Fig.1. Schematic diagram of experimental layout [1]

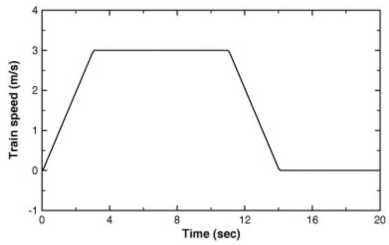

Fig.2. Schedule of train run [1]

2.2 Numerical simulation set up

In the current study, we perform a numerical simulation based on the same geometric configurations and train-run schedule that used by Kim and Kim [1] in their experimental study.

The computational models are constructed and solved using the CFD code FLUENT 6.3.26. The governing equations for airflow are continuity equation, Reynolds-averaged Navier-Stokes equations, energy equation and equation of state. The widely used standard κ-ε model is adopted for turbulence closure.

The governing equations are discretized on finite-volumes. The PISO algorithm is adopted to solve the pressure-velocity coupling. The first-order upwind scheme is employed for discretization of the physical parameters except for the pressure corrective equation. As for the pressure corrective equation, the PRESTO! Scheme [3] is used. For unsteady analysis, the time derivatives are discretized using the first-order implicit scheme. The computations are performed using the pressure-based solver.

For the numerical method for modeling the moving boundaries of the train, the dynamic mesh model which is available in FLUENT is adopted. With the dynamic layering method layers of cells adjacent to a moving boundary can be added or removed based on the height of the layer adjacent to the moving surface. Therefore, the update of the volume mesh is handled automatically by FLUENT at each time step during the process of calculation based on the new position of the train: the layer of cells adjacent to the front surface of a train is merged with the layer of cells next to it, while the layer of cells adjacent to the rear surface of train is split into two layers to make another layer of cells.

In the current calculation, the time step is set to be 0.05s.The grid distance in tunnel longitudinal direction (*X* axis) is set to be 0.03m. In the cross-section of tunnel (i.e., the *Y-Z* plane), the grid distance is set to be 0.02m in both *Y* and *Z* directions except for the narrow gap between the train passage zone and the tunnel walls, where the grid points are finely distributed. Initially, the entire computational domain consists of 334,400 hexahedra cells.

3. Numerical results of the current study

Figs.3-6 show the comparison between the numerical and experimental results for the variations of pressure coefficient C_p with time at each point of the four pressure transducers. Here, the results numerically obtained by Kim and Kim [1] are also presented for comparison. From these figures, it can be seen clearly that the present numerical results for C_p variations are in good agreement with the experimental data through the whole time period (0-20s). The qualitative behaviors of numerical results obtained by Kim and Kim [1] agree well with those of the experimental data. However, there are large discrepancies in C_p variations during the period of constant train speed after train passes the transducers.

Figs.7 and 8 show a comparison of u/U_{T_MAX} (Here, U_{T_MAX} is the maximum velocity of the train) variations between experimental and numerical results at each position of the two velocity transducers. It is found from these two figures that the present numerical results for u/U_{T_MAX} variations agree reasonably well with the experimental data although there is a relatively large difference between the numerical and experimental results when the train moves at a constant speed.

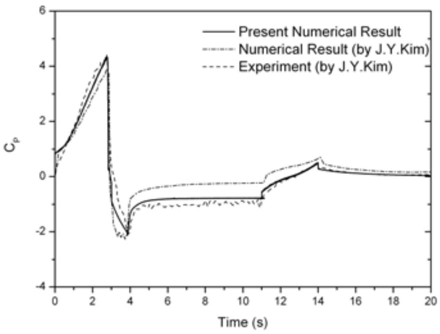

Fig.3 Comparison of C_p variations between experimental and numerical results at position of PT1.

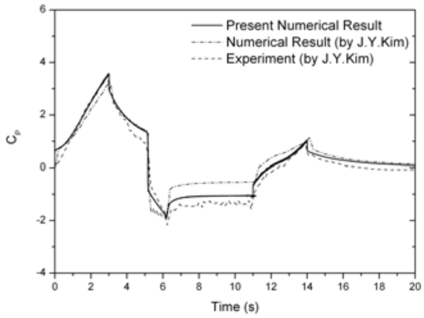

Fig.4 Comparison of C_p variations between experimental and numerical results at position of PT2.

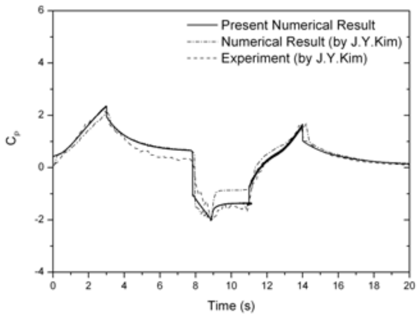

Fig.5 Comparison of C_p variations between experimental and numerical results at position of PT3.

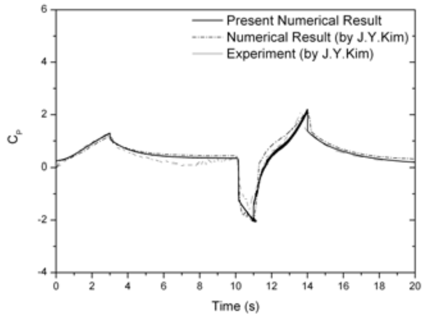

Fig.6 Comparison of C_p variations between experimental and numerical results at position of PT4.

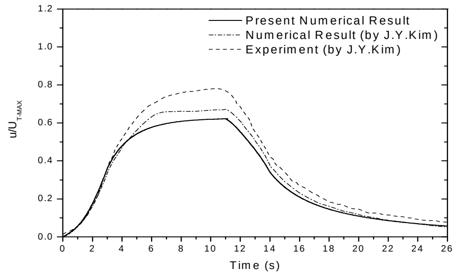

Fig. 7 Comparison of u/U_{T_MAX} variations between experimental and numerical results at position of VT1.

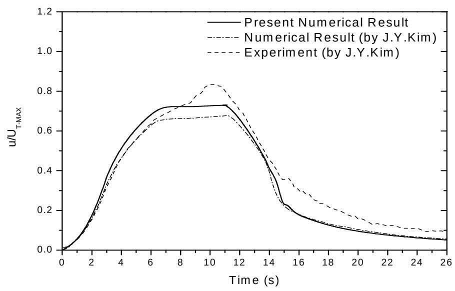

Fig. 8 Comparison of u/U_{T_MAX} variations between experimental and numerical results at position of VT2.

4. Conclusions

A three-dimensional numerical simulation on train-induced unsteady airflow is performed using the dynamic layering method for an experimental model tunnel. The numerical results are in good agreement with the experimental data, and it is shown that the dynamic layering method is applicable to modeling the train-induced airflow in a subway tunnel.

This project is supported by Seoul R&BD Program (CS070160).

References

1. J.Y. Kim and K.Y. Kim, “Experimental and numerical analyses of train-induced unsteady tunnel flow in subway”, Tunnelling and Underground Space Technology, Vol.22, pp.166-172, 2007.
2. Udaykumar, H.S., Mittal, R., Rampunggoon, P., Khanna, A., “ A sharp interface Cartesian grid method for simulating flows with complex moving boundaries”, Journal of Computational Physics, Vol.174, pp.345-380, 2001.
3. V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski and C.G. Speziale, “Development of turbulence models for shear flows by a double expansion technique”, Phys. Fluids A, Vol.47, pp. 1510–1520, 1992.