VALIDATION OF NUMERICAL SIMULATION SYSTEM FOR GAS DIFFUSION IN URBAN AREA

Ryohji Ohba, Akinori Kouchi, Tomohiro Hara, Jiro Yoneda
Mitsubishi Heavy Industries, Nagasaki R&D Center, Japan
Shinsuke Kato
Tokyo University, Institute of Industrial Science

ABSTRACT: We validated a gas diffusion model in urban area, using meso-scale meteorological model, RAMS and its dispersion model HYPACT with wind tunnel data observed by University of Hamburg and field data observed in Oklahoma City, 2003. RAMS has an optional scheme to simulate a building from a volume of fraction according to building shapes inside each mesh. However it takes much computational time, if we apply it in fine mesh scale. We developed a new computing scheme using RAMS code in order to simulate an effect of building complex with more fine mesh of few meters and high speed computing time of few minutes by a conventional personal computer. This computing scheme, first, calculates a steady flow in fine mesh for 16 wind directions, and stores them as a database of airflow. In this procedure, the shapes of buildings are extracted from digital map database. Next, we calculate an actual unsteady flow in wide mesh and nudging a linear combination of the database for 16 wind directions at the boundary in fine mesh domain by the Least Mean Square Method (LMSM). This simulator can predict concentration of hazardous gas with a few meters mesh size for a few minutes by a conventional PC.

1 INRODUCTION

The Ministry of Education, Culture, Sports, Science and Technology of Japan (MEXT) commenced a national project for urban safety in 2007. As a part of this project, MEXT assigned a research consortium consisting of the University of Tokyo, the National Institute for Advanced Industrial Science and Technology (AIST), Mitsubishi Heavy Industries (MHI) and AdvanceSoft, Corp. to develop a new hazard prediction and mitigation simulation system for CBR(Chemical, Bio and Radioactive materials) attacks. This simulation system is intended for use by first respondents to attacks such as municipal fire protection and police agencies. Because the system needs to be able to simulate gas diffusion with minimal computational time on a conventional personal computer (PC), we developed a new calculation scheme named "Database Computing System of Air Flow (DCSAF)". The similar kind of database computing system is developed by Patnaik and Boris (2006)[1] for gas diffusion pattern.

First, the present paper introduces an outline of our developing system and MEXT project. Next, validation results with wind tunnel data for a rectangular obstacle measured by University of Hamburg (http://www.mi.uni-hamburg.de/Data-Sets.432.0.html) and wind tunnel data of Suidobashi by Kato et al.(2009)[2].

2 SIMULATION SYSTEM

2.1 System description.

The developed simulation system, called MEASURES3, consists of an airflow database, a dispersion model, and a damage prediction model (Fig. 2.1-1). Meteorological data can be loaded onto the system by the user directly or through the Internet. The databases of the air-flow, topography, and population are pre-installed on the system. Less than 15 minutes of computational time are required for 12 hours of simulation. The output of simulations such as wind vectors, gas concentration, and casualty counts for every 10 minutes of simulated time are displayed on the screen, and are thus available to the users during and after the simulation.

2.2 Meteorological model

Most meteorological models are available in the public domain as open source codes. Examples of such models are RAMS, WRF, and MM5. To simulate airflow around building complexes with a mesh size of a few meters, the RAMS (Regional Atmospheric Modeling System) code developed by Pielke et al. of Colorado State University (http://www.atmet.com/) was modified. First, the two-dimensional turbulence model by Mellor and Yamada (1974) [3] was replaced by a three-dimensional turbulence model as in Castelli (2006) [4] to simulate complex airflow around buildings and terrain.

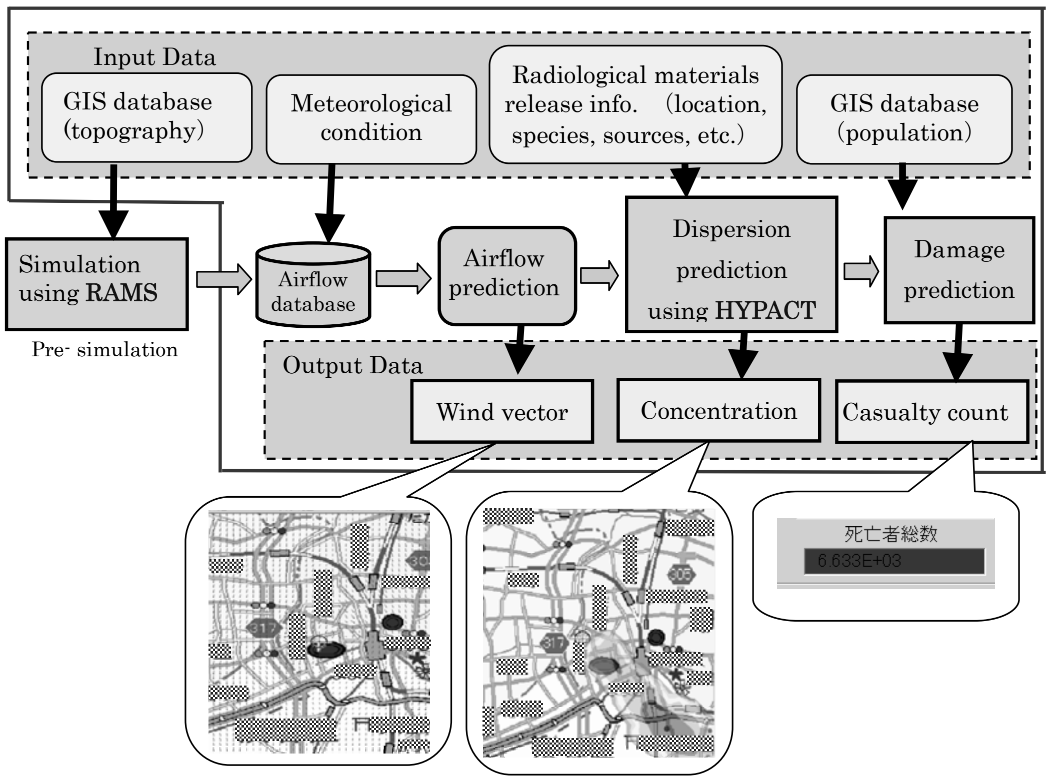


Figure 2.1-1: Structure of the advanced edition of MEASURES

Subsequently, the high speed computing scheme, Database Computing System of Air Flow (DCSAF) was developed by Ohba (2008) [5]. The computing system of DCSAF can be summarized as follows:

1) Steady-state airflow is calculated for a location of interest under various atmospheric conditions: a total of 48 conditions were generated by combining 16 wind directions and 3 atmospheric stabilities.

2) A time series of airflow is generated for the location of interest. In this procedure, the 3-dimensional wind data from the database of 48 atmospheric conditions are interpolated at each time step for the wind direction and the atmospheric stability observed at the location at that time step (Fig. 2.2-2).

Because computational time is inversely proportional to the mesh size, simulation with a small mesh size requires significant computational time. However, with the use of DCSAF, the computational time is reduced by a factor of approximately 100 compared to that by a conventional model (Table 2.2-1).

Table 2.2-1: Time required to compute one simulated hour using a PC. Times given are those required to calculate the large-scale grids, the small-scale grids, and the large- and small-scale grids combined, respectively.

Model	100 m <	100 m >	Total
Present model	Few min.	Few sec	Few min.
Conventional	10 min.	200 min.	Few hrs

2.3 Atmospheric diffusion model

The HYPACT (HYbrid PArticle and Concentration Transport) code is an atmospheric diffusion code that can be coupled to RAMS. This code is based on a Lagrangian particle model that satisfies mass conservation in complex airflow and can adopt the finite difference method at large distances downwind to reduce computational time.

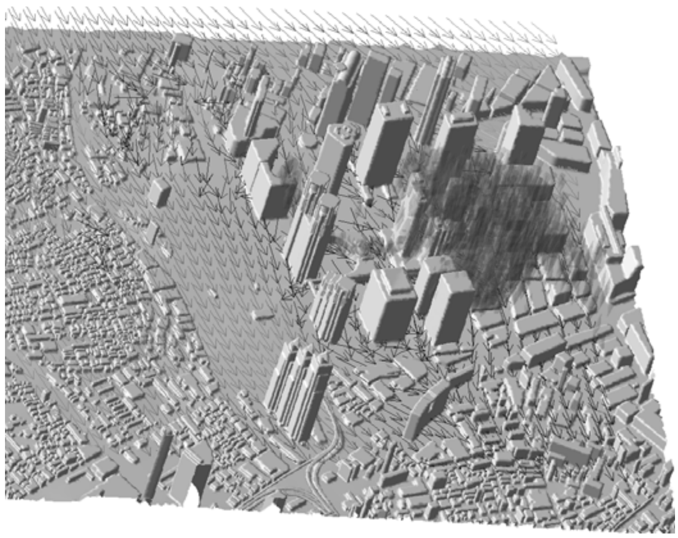


Figure 2.3-1: Simulated results of airflow and gas diffusion around actual buildings in Tokyo metropolis

3 VALIDATION STUDY OF MEASURES3 SYSTEM

3.1 Air flow and gas diffusion around a rectangular obstacle in wind tunnel

The present study explores the capability of the RAMS and HYPACT numerical models to reproduce experimental wind flows and concentration patterns around single block buildings. It is essential that numerical models are fully validated using quality assured and fully documented experimental datasets. To this scope, a wall-mounted cube, representing the simplest structural unit of urban canopies, is used to test the model. Despite its simplicity, wall-mounted cube within a fully-developed turbulent wind flow can reproduce the most salient features of wind flow around real buildings, i.e. sharp pressure gradients, multiple separation and re-attachment regions. For this reason, wall-mounted cubes have been commonly used as a benchmark for the evaluation of advanced numerical models.

In addition to HYPACT, the semi-empirical model ADMS, developed by CERC, has been used to produce mean concentration field to be compared with the wind tunnel data and the outputs of HYPACT.

The Model Evaluation Guidance and Protocol Document published by COST-732 has been used for selecting appropriate metrics to help assisting the statistical interpretation of the results.

3.1.1 Wind tunnel data of mean wind flow and TKE

The wind tunnel experiments were carried out in the BLASIUS wind tunnel at the Meteorological Institute of the University of Hamburg. The data sets used in this study is that relative to mean flow and turbulence around an isolated obstacle. The data sets are categorized as CEDVAL A1-1 and are available for download at the web site http://www.mi.uni-hamburg.de/Data-Sets.432.0.html.

In Figure 3.1.1-1, a cross sectional view of the mean flow field is shown for the wind tunnel experiment and the simulation model. The flow separates at the upwind corner of the obstacle, resulting in an accelerated vertical flow in the impingement region.

a) Wind tunnel experiment b) simulation model

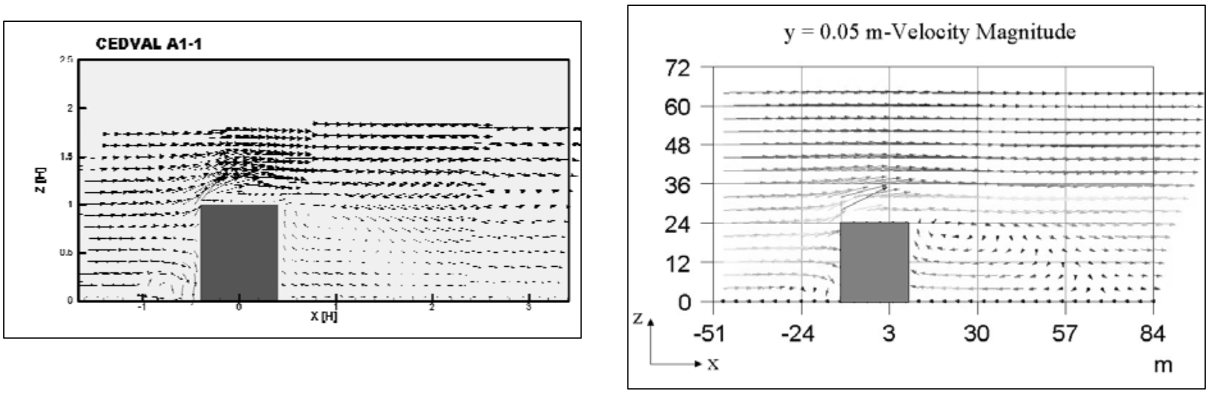


Figure 3.1.1-1: Vertical distribution of wind vectors around an obstacle

In order to assess the capability of the model to mimic physically realistic flow field, mean velocity components and TKE (Total Kinetic Energy) profiles computed by the model have been compared against wind tunnel data at several positions. Results are reported in the following Figure 5a- as, relatively to mean horizontal velocity (u), and TKE.

The RAMS model overestimates the TKE at up and downstream positions (Figure3.1.1-2, 4). Both the model and the wind tunnel, predict a TKE peak at roof level, for z/H ≈ 1, but the model underestimates horizontal velocity at downstream positions. This will have significant implications in the prediction of gas concentration in the wake of the obstacle (see Section 3.1.2).

a) Horizontal velocity b) TKE

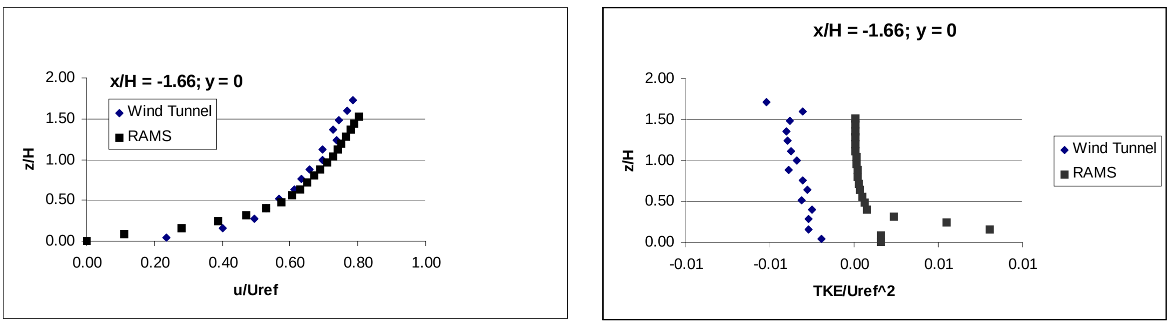


Figure 3.1.1-2: Horizontal velocity and TKE at upwind side of an obstacle

. a) Horizontal velocity b) TKE

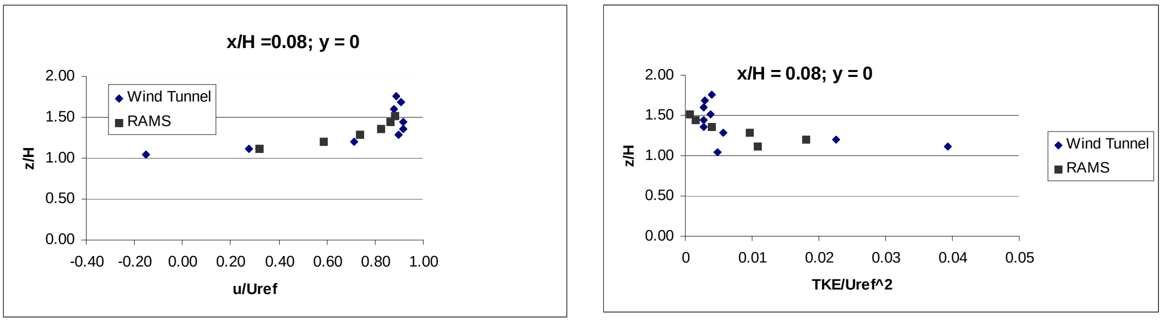


Figure 3.1.1-3: Horizontal velocity and TKE over the roof of an obstacle

. a) Horizontal velocity b) TKE

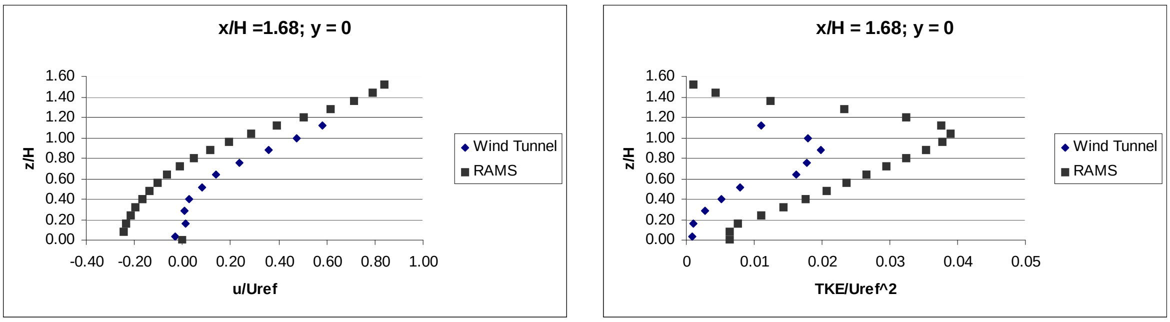


Figure 3.1.1-4: Horizontal velocity and TKE behind an obstacle

3.1.2 Gas concentration

In the wind tunnel experiment classified as CEDVAL A1-5, four area sources were placed at the corner of the obstacle's leeward wall with the ground (at z = 0.0 m). Results are given in terms of normalised concentrations K (Eq. 5) at several longitudinal (y-direction) and vertical (z-direction) sections.

The release of a passive tracer (not specified) was simulated in the experiment, with an exit velocity of approximately 0.015 m s^{-1}. Data for *K* are provided with an uncertainty of ± 0.018.

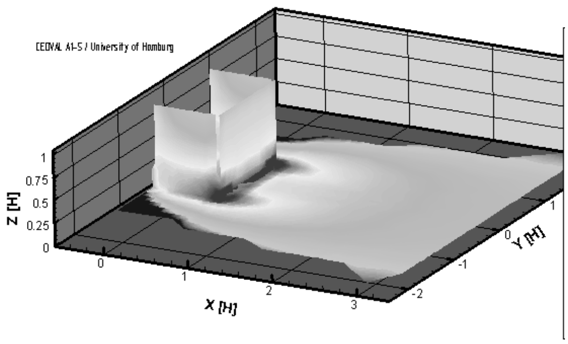


Figure 3.1.2-1 Contours of the scaled concentration K for z = 2.0 m observed in the wind tunnel (from: http://www.mi.uni-hamburg.de/Category-A.628.0.html).

Mean concentration results from CEDVAL are expressed as non-dimensional concentration K, with

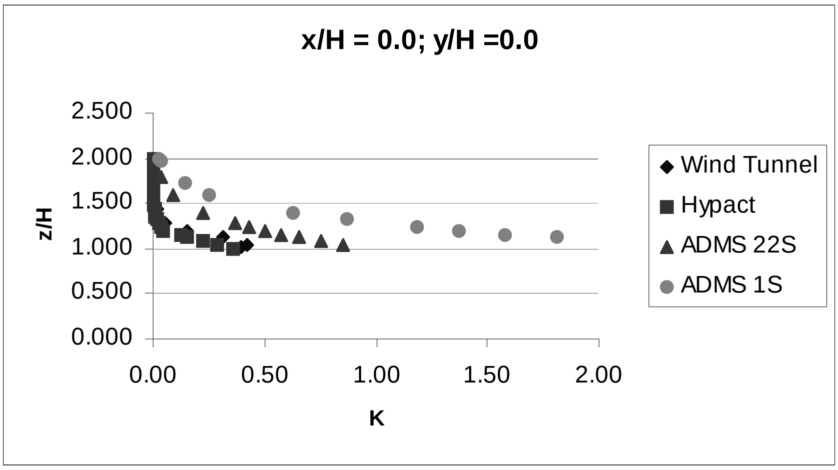
Eq. 1)
$$K = \frac{Conc\, U_{ref}\ H^2}{Q_C}$$

in which:
Conc is the measured concentration (µg m-3);

Uref is the mean horizontal velocity at the top of the boundary layer (= 5.86 m s-1)
H is the reference height (= 25 m)
QC is the flow emission rate of pollutant from the point source (μg s-1).

The graphs in Fig 3.1.2-2 show the comparison between HYPACT,ADMS and wind tunnel data at several position downstream the obstacle, for the vertical plane y/H = 0.0. The result of HYPACT agrees well with the wind tunnel data. ADMS was developed by CERC (Cambridge Environmental Research Consultant,http://www.cerc.co.uk/) to calculate the concentration patterns considering the effect of terrain. ADMS 4 includes a building module to model the effects of buildings on the dispersion of pollutants. ADMS internally divides the flow field into two parts: a recalculating region (or cavity), and a diminishing turbulent wake downstream. Concentrations within the cavity are uniform and based on the fraction of the release that is entrained.

a) Over the obstacle



b) Behind the obstacle

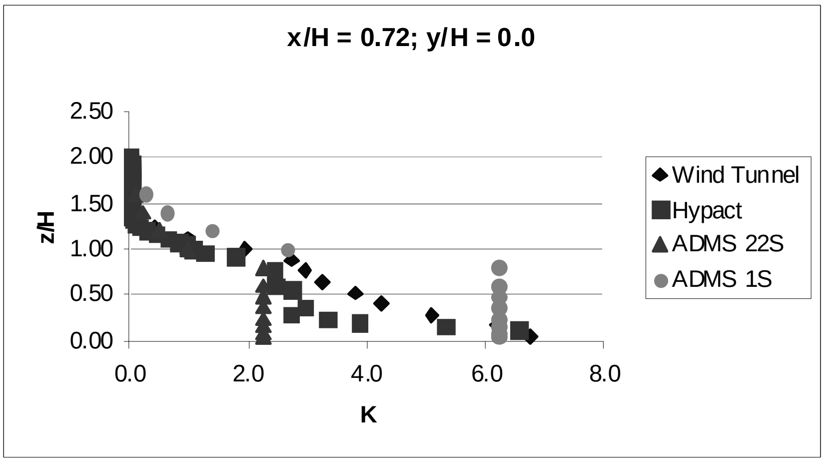


Figure 3.1.2-2. Comparison for K predicted by wind tunnel data, HYPACT, and ADMS (1 source and 8 sources) for several positions downstream the obstacle

3.2 Gas diffusion over an actual urban area

Gas concentration was measured for an actual urban area near Suidobashi in Tokyo, as shown in Fig. 3.2-1, using the wind tunnel of Tokyo University (2009, Kato et al.)[2].
3D view of particle distribution around building complex was calculated by RAMS&HYPACT code, as shown in Fig. 3.2-2. It can be seen from this figure that building complex has much effects on gas diffusion in urban area.

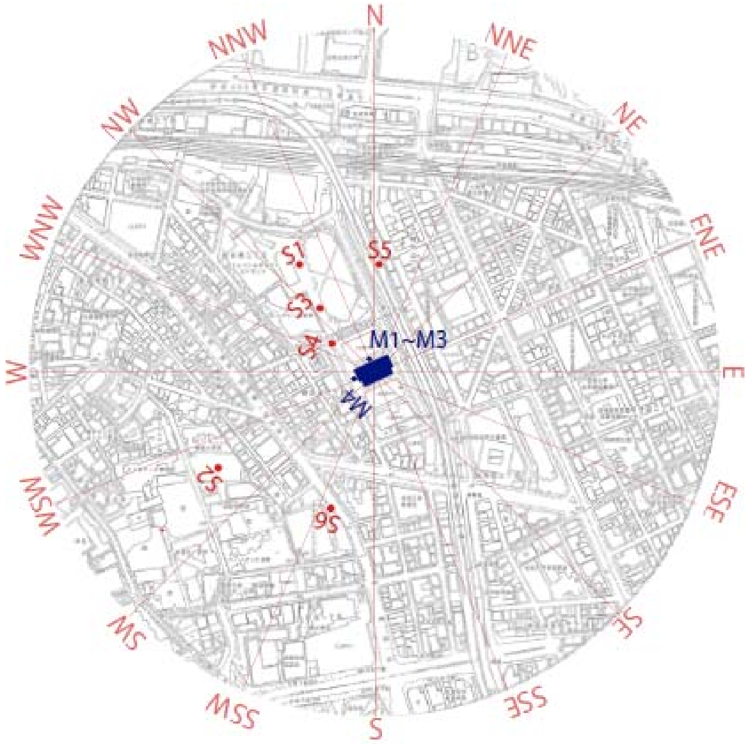


Fig. 3.1-1 Region, measuring point (M1 to M4) and source position (S1 to S6) in scaled model of Suidobashi

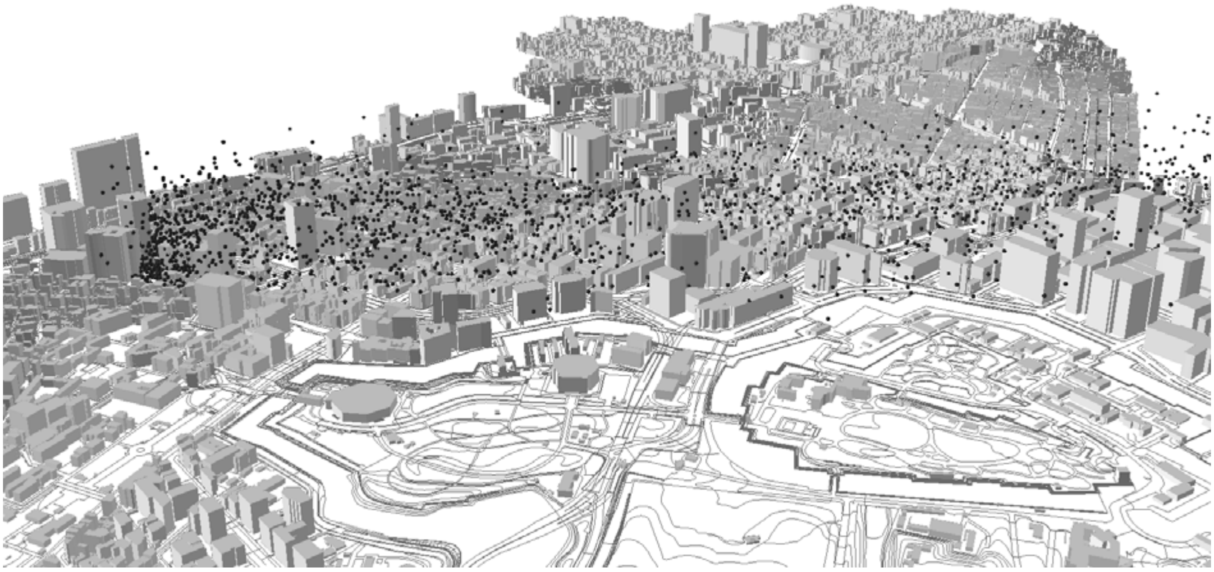


Fig. 3.2-2 3D view of particle distribution around building complex in Suidobashi, Tokyo

Simulated results were compared with wind tunnel data for averaged steady value of time series in Fig. 3.2-3. Although the agreement of quantitative value is not good, the relative difference for each measuring point is the same tendency for both data. The reason of underestimation by simulated results is supposed to be caused by the blocking effect of many low buildings with less than 50m height omitted for the numerical simulation. We are now making an improvement to simulate the omitted low buildings as a canopy layer of air flow. The canopy layer has an effect to decrease wind velocity near the ground and to increase gas concentration.

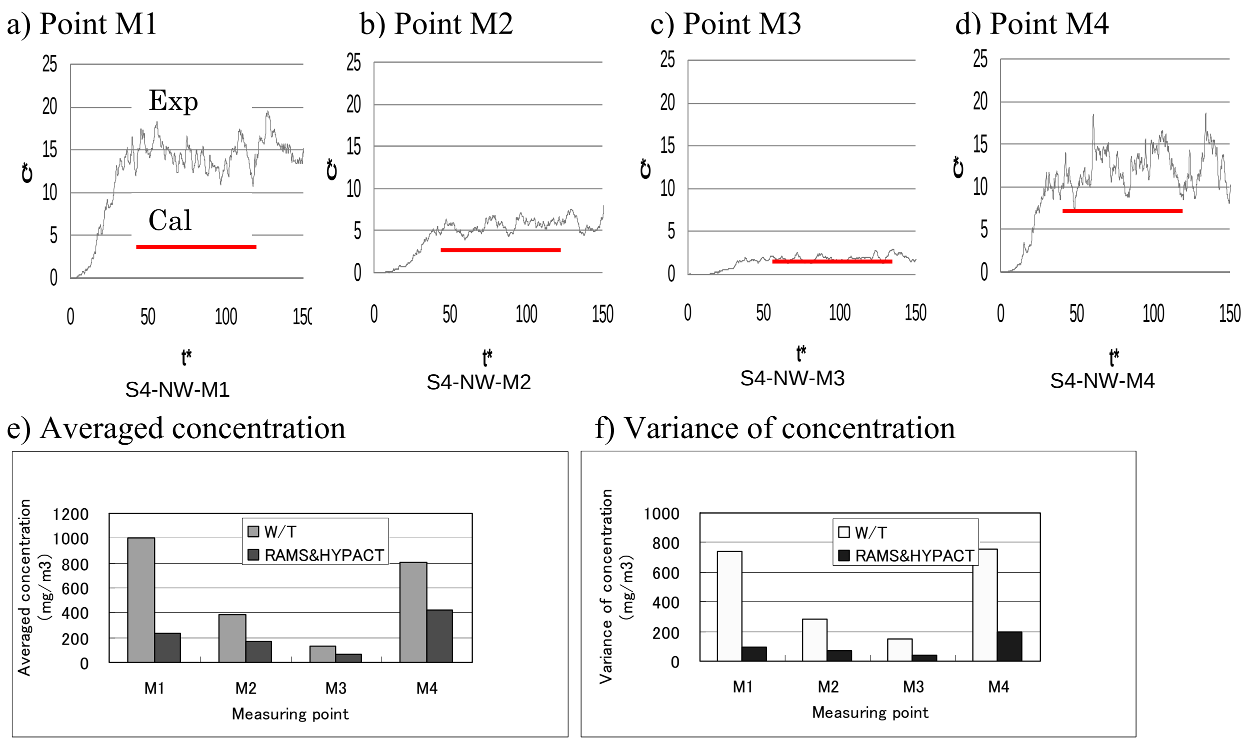


Fig. 3.2-3 Comparison of simulated and experimental data for concentration of time series under the wind condition of NW and the release point of S4

4 *CONCLUSIONS*

It was found from this study that

1) RAMS&HYPACT codes can simulate well the air flow and gas diffusion around a rectangular building.
2) There are some discrepancies between the simulated and measured results for gas concentration over an actual urban area in Tokyo, but relative difference at each measuring point is in almost the same for both results.

5 *AKNOLEDGEMENTS*

This study is conducted under the sponsorship by Ministry of Education and Science in Japan, and in corporation with Tokyo University. The validation study for the rectangular building was jointly conducted with CERC.

6 *REFERENCES*

1 Patnaik, G and Boris, J. P., Fast and Accurate CBR Defense: Bringing HPC to the War fighter and First Responder, Proceedings of 44^{th} AIAA Aerospace Science Meeting and Exhibit, Reno, Nevada, 2006
2 Kato S. et al., Wind Tunnel Experiment to Validate Numerical Simulation for Unsteady Gas Dispersion in Urban Area, Proceedings of PHYSMOD2009, VKI, Sint-Genesius-Rode, Belgium, 2009(to be submitted)
3 Mellor, G.L., Yamada, T., 1974. A hierarchy of turbulence closure models for planetary boundary layers. Journal of the Atmospheric Sciences 31, 1791–1806.
4 Castelli, T. S. et al., Validation studies of turbulence closure schemes for high resolutions in mesoscale meteorological models, Atmospheric Environment 40, 2006
5 Ohba, R. et al., Development of Hazard Prediction System for Intentional Attacks in Urban Areas, Urban Safety of Mega City in Asia, 2009, Beijing China