

## ATTEMPT TO MAKE GUIDELINE TO USE CFD MODEL FOR ATMOSPHERIC ENVIRONMENTAL ASSESSMENT IN URBAN AREA IN JAPAN

Hiroaki Kondo\*, Kenji Horiuchi\*\*, Yoneko Hirano\*\*\*, Norihisa Maeyama\*\*\*\*, Kazuhiko Ogata†, Satoru Iizuka††, Tateki, Mizuno\*\*\*\*

\*National Institute of Advanced Industrial Science and Technology (AIST), Tsukuba, Japan;

\*\*Institute Research & Development Co., Ltd, Fujisawa, Japan,

\*\*\*Environmental Technical Laboratory Ltd, Tokyo, Japan, \*\*\*\*Japan Weather Association, Tokyo, Japan,

†SUURI-KEIKAKU Co., Ltd., Tokyo Japan, ††Nagoya University, Nagoya, Japan

### Abstract

There have been plenty of CFD (Computational Fluid Dynamics) models and their applications to atmospheric environmental issues. CFD models are capable of calculating many complicated situations, such as the flow and dispersion around complicated urban complex, which are not well described with the traditional Gaussian models. However, many parts in the commercial CFD software are a kind of 'black box' for most of the users and CFD results often depend on some factors, such as grid interval, computational schemes and so on. A voluntary committee in the Japan Society for Atmospheric Environment started to make a guideline for the application of CFD models to environmental assessment in the atmosphere. The committee mainly investigated the results from CFD models using the standard  $k-\varepsilon$  turbulence closure, which is widely used in the world. At first, the committee tried to determine the condition that a variety of software gave the same results for the diffusion from a point source above a flat surface. Then the results of CFD models were compared for the diffusion around a surface mounted cube with those of wind tunnel experiment. Some knowledge from these activities is presented.

**Key words:** DiMCFD, Environmental Assessment, Guideline

### 1. INTRODUCTION

Many commercial CFD softwares for general purposes have been developed in recent decades associated with the development of computer technology, and it becomes more frequent for them to be applied to practical assessment in urban atmospheric environment. This Diffusion Model based on Computational Fluid Dynamics (DiMCFD) may be capable of estimating concentration around a building with complicated shape and around complex terrain, which was not able to be estimated with conventional Gaussian models. However users of these softwares are not always specialist of CFD models, and there are many problems and know-how in the practical application. A voluntary committee to investigate applicability of DiMCFD models to environmental assessment in the atmosphere was established in Sep., 2006 in the Japan Society for Atmospheric Environment, and an interim report was published in this May.

The committee tried the following six steps. The members of the committee ran their softwares by themselves and inter-compared the results.

- 1) Diffusion from a point source above a flat surface was calculated. Aerodynamic roughness and inlet wind profile are given.
- 2) The height of the point source was fixed at 30 m above the surface, and inlet profile for  $k$  and  $\varepsilon$  were fixed.
- 3) Grid intervals were specified and calculation was performed on smooth surface.
- 4) The same calculation was performed on rough surface.
- 5) The results of DiMCFD for a point source on the flat surface were compared with those of wind tunnel experiment.
- 6) The results of DiMCFD for a point source set in the upstream side of a surface mounted cube were compared with those of tunnel experiment.

### 2. MODELS

All the DiMCFD models participated in this inter-comparison were RANS (Reynolds Averaged Navier-Stokes equations) model with standard  $k-\varepsilon$  turbulent closure (Launder and Spalding, 1974). They used either of Eulerian or Lagrangean method for material transportation (Table 1).

### 3. RESULTS

The results of each step are briefly summarized below.

#### 3.1. STEP I

Only inlet wind profile and surface roughness were fixed and diffusion from a point source near the surface (Its location was not strictly specified.) was calculated. The participants set grid interval according to their experience (Table 2). The results showed that wind velocity profile and its variation toward downstream were similar among the models; however profiles of turbulent kinetic energy  $k$  and dissipation rate  $\varepsilon$  were much different. The dispersion parameters (width of dispersion) near the surface calculated from the concentration pattern were much different as well.

Table 1 Softwares used in the present inter-comparison

Symbol in this paper	Software name	Vender	Mass transport: Euler or Lagrange
AK	α-FLOW	Mizuho Information & Research Institute	E
IH-1	STAR-LT	CD-Adapco Japan	E
IH-2	Open FOAM	OpenCFD	E
JM	In-house development	JWA	L
KH	STAR-LT	CD-Adapco Japan	E
KH +	STAR-CCM+	CD-Adapco Japan	E
KK	FLUENT	FLUENT ASIA PACIFIC	E
NT	STREAM	SOFTWARE CRADLE	E
SO	Phoenics	CHAM-Japan	E

Table 2 Grid intervals specified by each participant (example)

Symbol	stream wise		span wise		vertical	
	grid interval	domain	grid interval	domain	grid interval	domain
	minimum ~		minimum ~		minimum ~	
IH-1	1m ~	2000m	1m ~	1000m	0.2m ~	1000m
JM	10m ~	1500m	1m ~	1000m	1m ~	200m
AK	20m(fixed)	2000m	10m(fixed)	300m	10m(fixed)	200m
SO	1m ~	1030m	1m ~	300m	0.5m ~	210m

### 3.2. STEP II

The members discussed the reason for the dispersive results in STEP I and decided that they aimed to obtain similar results at first. Then inlet profiles of  $k$ ,  $\varepsilon$  and source height (30 m) were fixed. The results showed that wind velocity profile and its variation towards downstream were similar among the models as well as those in STEP I; however those of  $k$  and  $\varepsilon$  were still dispersive.

### 3.3. STEP III

Then members checked again the parameters used in their DiMCFD models, for example, density of the fluid, molecular viscosity, thermal diffusivity, and so on, which were mostly given as default values. In this checking process, it was found that one model used the parameters not for air but for water. Then, wall function and parameters used in it were checked and the wall function for smooth surface was chosen for the bottom surface boundary. Finally, all the grid intervals and calculating domain were unified. After these restrictions, the dispersion of the results among the models became very small.

### 3.4. STEP IV

The bottom boundary was changed from smooth to rough. Then the members noticed that the turbulence kinetic energy from inlet boundary and that generated on the rough surface were not balanced along the long distance from the inlet boundary, and that turbulence in the boundary layer was not in equilibrium along the flow. Surface roughness, wind profile and boundary layer height should be linked in order to realize quasi equilibrium state within short distance from the inlet boundary. Then the inlet boundary conditions were changed according to the recommendation of the Architecture Institute of Japan (AIJ)'s guidebook (2007). These conditions are as follows.

$$U(z) = U_s \left( \frac{z}{z_s} \right)^\alpha$$

$$I(z) = 0.1 \left( \frac{z}{z_G} \right)^{(-\alpha-0.05)}$$

$$k(z) = (I(z)U(z))^2$$

$$\varepsilon(z) = C_\mu^{1/2} k(z) \frac{U_s}{z_s} \alpha \left( \frac{z}{z_s} \right)^{(\alpha-1)}$$

where,  $U_s$ ,  $z_s$ ,  $z_G$ ,  $\alpha$  and  $C_\mu$  are ensemble averaged wind velocity at  $z_s$ , reference height, boundary layer height, power for the power law of wind profile, and a constant equal to 0.09, respectively. In this inter-comparison,  $\alpha=1/7$ ,  $U_s=10$  ( $\text{ms}^{-1}$ ),  $z_s=z_G=100\text{m}$  were used.

### 3.5. STEP V

Since the members confirmed all the models were able to produce similar results, then, DiMCFD results were compared with those of wind tunnel experiment in the next step. The wind tunnel experiment for the first comparison was diffusion from a point source at 5 cm above the floor of the test section. Wind profile with 1/7 power law was formed with turbulence generator in the wind tunnel. Profiles of wind velocity, turbulent kinetic energy and dissipation rate obtained from wind tunnel experiment were used as inlet boundary conditions of DiMCFD models. The scale of the DiMCFD models was the same as the scale of wind tunnel and no similarity relation was used. A  $\Gamma$  shaped stack with 4mm diameter was used for the source. Figure 1 shows the comparison of the concentration profile along the centerline of the plume obtained from wind tunnel (pink) with those from one of DiMCFD result (blue) with turbulent Schmidt number ( $S_{ct}$ ) of 0.9. They agreed with each other at the location farther than 1.5 m downstream from the source; however DiMCFD result more diffused near the source. If this discrepancy was adjusted with  $S_{ct}$ , the required  $S_{ct}$  is approximately 4 for the case at 0.3 m downstream from the source. Lagrangean mass transport model did not show this excess of diffusion near the point source.

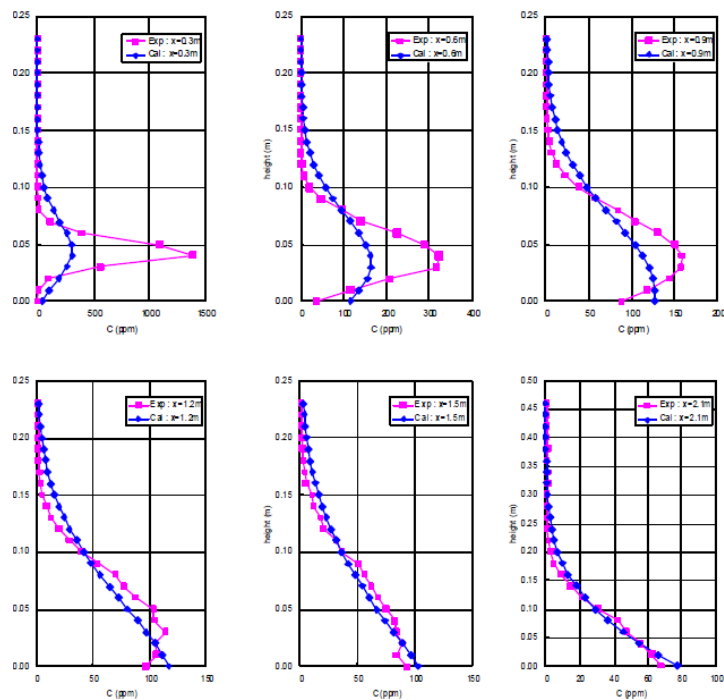


Fig.1 Comparison of concentration profile along the plume axis between IH-1 and wind tunnel experiment. Other Eulerian mass transport models gave similar result.

### 3.6. STEP VI

The results of DiMCFD were compared with those of wind tunnel experiment for the diffusion around a surface mounted cube, the height of which was 0.1 m (=H). A point source was set at 1.5H upstream of the center of the cube and at 0.5H above the floor of the wind tunnel. All the Eulerian DiMCFD models gave similar results to those of already published works with the standard  $k-\epsilon$  model for flow and turbulence, which does not always mean the DiMCFD results agreed with those from wind tunnel experiment.

All the results of concentration pattern obtained from DiMCFD with Eulerian mass transport were similar; however the difference between those of DiMCFD and wind tunnel experiment was remarkable at wake region of the cube. The difference of the concentration pattern near the surface was relatively small. It is difficult to say which transport model gave better results in comparison with the wind tunnel experiment. DiMCFD gave better results than those of improved Gaussian models such as METI-LIS (JEMAI, 2009), which includes down wash effect due to buildings.

## 4. DISCUSSION AND FUTURE WORK

Our activity was motivated by the fact that there was no guideline and consensus for the applicability of DiMCFD to environmental assessment in the atmosphere. The committee includes those who are not specialist for CFD models. Some of above results may have been already obvious for the specialists for CFD models. However applicability of DiMCFD model to practical assessment is different problem from the accuracy of DiMCFD models, and step by step approach here we taken was very much helpful to understanding DiMCFD for all the members. Similar activities to make guideline to use CFD models or wind tunnel experiment have been

conducted by AIJ (AIJ, 2007), Atomic Energy Society of Japan in Japan for their own problems and COST 732 in Europe (Franke et al., 2007). Their experiences are very much helpful for our understanding of DiMCFD.

Our tentative targets for next stage are as follows. (The order does not mean the level of importance)

- 1) How to calculate time average specified in air quality standard?
- 2) To what kind of problem should we apply DiMCFD?
- 3) To clarify the permissible range of error for the environmental assessment.
- 4) Applicability of DiMCFD to the problem affected by stability

At present we have not used field data yet. The difference of the results among the field experiment, wind tunnel experiment and DiMCFD is another important point to be discussed in future.

#### **Acknowledgement**

The wind tunnel experiment in the present study was supported by KAKENHI (19510035) and Ministry of Environment, Japan.

#### **Reference**

- Architecture Institute of Japan (AIJ), 2007. Guide book for CFD analysis for estimating wind environment in urban area. – Guidelines and database for validation -, Maruzen, Tokyo, p.207. (in Japanese)
- Franke, J., Hellsten, A., Schlünzen, H., Carissimo, B. (eds.), 2007. Best practice guideline for the CFD simulation of flows in the urban environment. COST Action 732, Quality assurance and improvement of microscale meteorological models. p.52.
- JEMAI, 2009. [http://www.jemai.or.jp/CACHE/tech\\_details\\_detailobj1816.cfm](http://www.jemai.or.jp/CACHE/tech_details_detailobj1816.cfm) (latest version, in Japanese)
- Launder, B. E., Spalding, D. B., 1974. The numerical computation of turbulent flows. *Comp. Method in Applied Mech. and Eng.* 3, 269-289.