J. of Korean Oil Chemists' Soc.,
Vol. 29, No. 1. March, 2012. 33~39


# Two-dimension Numerical Simulation of Stack Flue Gas Dispersion

Young-Koo Park† · Shi-Chang Wu*

†*Dept. of Env. Eng., Kangwon National University, Samcheok 245-711, Korea*
**Department of Environmental Science and Engineering,*
*Kyung Hee University, Gyeonggi-Do, 446-701, Korea*
*(Received January 12, 2012 ; Revised March 6, 2012 ; Accepted March 9, 2012)*

**Abstract** : A numerical simulation of plume from a stack into atmospheric cross flow is investigated using a two-dimension model. The simulation is based on the $\kappa \sim \varepsilon$ turbulence model and a finite volume method. In this paper, it mostly researches how the wind velocity affects the flue gas diffusion from an 80 m high stack. Wind velocity is one of the most important factors for flue gas diffusion. The plume shape size, the injection height, the NO pollutant distribution and the concentration at the near ground are presented with two kinds of wind velocities, 1 m/s and 5 m/s. It is found that large wind velocity is better for flue gas diffusion, it generates less downwash. Although the rise height is lower, the pollutant dilutes faster and more sufficient.

*Keywords : Numerical simulation, Atmospheric cross flow; Stack, Wind velocity; Flue gas, Plume diffusion.*

## 1. Introduction

Plumes and jets issued into cross flow have been the subject of intensive research. An important application of this type of flow is in the dispersion of pollutants from stacks of power generation and industrial sites. The entrainment of exhaust gases released by short stacks or rooftop vents into the wakes of buildings can result in ground-level pollutant concentrations. To help dispersion of emissions, the concept using tall chimneys is commonly adopted around the globe [5]Understanding the different phases of the mixing process of plumes and evaluating how the plumes diffuse and spread is a matter of considerable interest in environmental impact studies and in basic numerical simulation of plume dynamics. Numerical simulation method is also called Computational Fluid Dynamics (CFD). It is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The basic treatment softwares are Gambit and Fluent. CFD can be seen that it control the basic equations (mass, momentum, energy conservation) to simulate the flow. Through the numerical simulation, it can distribute the fundamental physical quantity (e.g. velocity, pressure and temperature) at different positions in complicated cases and

†Corresponding author
(E-mail : ygpark@kangwon.ac.kr)

the relationship with the time. Previous studies on plumes dispersion in the atmospheric environment are commonly discussed. When a contaminant is released into a cross-flow it mixes and dilutes with the ambient air as a result of complex turbulent processes. As a consequence, the concentration of the contaminant will vary in space and time even for a steady release rate[8]The downwash effects including enhanced dispersion in the wake, reduced plume rise due to streamline deflection and increased turbulence, and a continuous treatment of the near and far wakes [7].Sensitivity analyses of parametric performances of the flow were performed by changing the parameters including chimney height, relative humidity of atmosphere, and wind velocity in the simulation [10]. Many reasons cause the effects of plume diffusion, like flue gas injection velocity, the t e m p e r a t u r e $\frac{\partial(\rho T)}{\partial t} + \mathrm{div}(\rho uT) = \mathrm{div}\left(\frac{k}{C_p}\mathrm{grad}T\right) + S_T$ difference between flue gas temperature and ambient temperature, ambient wind velocity, atmospheric turbulent intensity, regional climate and so on. Here the wind velocity is the most important factor in the diffusion, so that it is discussed mostly below.

In order to research the flue gas diffusion in the centre plane deeply, two-dimension numerical simulation is adopted here. It can make a simpler geometry model and better quality meshes. The Research focuses on the trajectories of plume diffusion and pollutant diffusion in different wind velocities. The clear figures below can give the answers.

## 2. Models and boundary conditions

### 2.1. Mathematical model

In order to evaluate the performance of a plume and pollutant concentration from a stack in an atmospheric cross flow, a standard ~ ε turbulent model is adopted, where κ is the turbulence energy and ε is the rate of dissipation of κ. The κ ~ ε model is based on the eddy viscosity hypothesis. The evolution of a turbulent and buoyant plume can be described by the time-averaged forms of the differential equations of mass, momentum and energy in two-dimensional form. In addition, an equation of state links the density to the temperature[4].

Mass:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \qquad (1)$$

Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \mathrm{div}(\rho uu) = \mathrm{div}(\mu\, \mathrm{grad}u) - \frac{\partial p}{\partial x} + S_u \qquad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + \mathrm{div}(\rho vu) = \mathrm{div}(\mu\, \mathrm{grad}v) - \frac{\partial p}{\partial x} + S_v \qquad (3)$$

Energy:

$$\frac{\partial(\rho T)}{\partial t} + \mathrm{div}(\rho uT) = \mathrm{div}\left(\frac{k}{C_p}\mathrm{grad}T\right) + S_T \qquad (4)$$

State:

$$P = \rho RT;\ e = C_v T \qquad (5)$$

The numerical code uses the finite volume method. In the turbulent model, segregated solver related to the discretization of the momentum and continuity equations and their solutions by means of the segregated solver are addressed. They are most easily described by considering the steady-state momentum and continuity equations in integral forms. The solver bases on pressure, using pressure modification method to discretize. There are different methods for pressure equations including a method known as semi-implicit method for pressure linked equation(SIMPLE) [9].The algorithm of SIMPLE employs a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. Since the equations are not solved exactly, there is a certain level of residual for each computational cell for each variable. In the numerical model, air follows the incompressible ideal gas law because the Mach number of air in the case is less than

0.3. The Mach number is a dimensionless value useful for analyzing fluid flow dynamics problems which compressible is a significant factor. It is defined that:
Mack number:

$$M = \frac{v}{c} \quad (6)$$

Here c is the sound velocity, v is fluid flow velocity. In the case, the fluid velocity is 20.5 m/s, so M just equals to 0.06. If the Mach number is less than 0.3, the fluid is approximate thought as incompressible. Because of the large temperature difference between the source and ambient area, the Boussinesq approximation (buoyancy effect) was not adopted here [2].

### 2.2. Computational models

The geometry model which is made with Gambit is shown in Fig. 1.

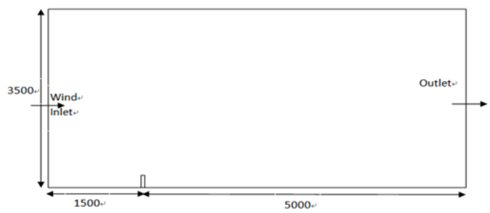



Fig. 1. Computational domain.

The proportion of the model is 1:1000 to the reality. The unit in the Fluent is millimeter. The total reality calculation area is 6500 m long and 3500 m high. The stack height is 80 m and the diameter is 3.5 m, which is located at 1500 m downstream of the cross wind inlet. The height of the domain is made high enough that the plume development is not influenced by the boundary conditions at the top, and hence symmetry condition can be assumed there.

The model grids comprise triangular or quadrilateral cells in 2D. Here the later one is adopted. As the regulation of the model geometry, structured grid is used here. It makes the mesh good quality. The interval of cells is 20, and the total mesh number is 57366.

### 2.3. Boundary conditions

Boundary conditions specify the flow and thermal variables on the boundaries of the physical model. They are, therefore, a critical component of simulations and it is important that they are specified appropriately.

The cross wind inlet sets as the velocity inlet which is usage for incompressible flow. Here air can be assumed to incompressible flow. The reason has been explained above. The wind shear is neglected, thus it is used with a uniform wind velocity. The cross wind velocity is set to $U_x$=1m/s and 5m/s two kinds, the turbulence energy($\kappa$ = 1 $m^2/s^2$) and its dissipation rate(=1 $m^2/s^3$) are adopted the default values for typical neutral stability conditions. The ambient temperature, T is set to 20°C. In the case, the vertical velocity component, v, is set to 20.5 m/s at the stack velocity inlet and $\kappa$ and $\varepsilon$ are also using the default values. The temperature of the plume from the stack inlet is set to 115°C above the ambient temperature. Due to consideration of the pollutant diffusion, the species transport equation is open here. The flue gas contains 60 ppm NO, 15.6 % $O_2$, 5.79% $H_2O$ and the left is $N_2$. The floor of the flow domain and the walls of the stack are treated as the wall boundary. The flow cannot pass by the wall. It is set adiabatic (zero heat flux) condition, stationary wall and no slip, therefore the temperature of the plume in the stack remained constant up to the point of releasing to the atmosphere. A Neumann boundary condition is imposed at the outlet. Outflow boundary condition is that a zero diffusion flux for all flow variables and an overall mass balance correction. The condition at the outlet imposes no restriction in any variables, has no impact on the upstream flow, but assumes a zero gradient for the streamwise velocity which is equivalent to a fully developed flow with v component of the velocity set to zero.

The top of the flow domain is treated as the symmetry condition. It assumes a zero flux of all quantities across the symmetry boundary. There is no convective flux across the symmetry boundary, and the normal velocity component is thus zero and the normal gradient of all flow variables are thus zero, too.

## 3. Results and discussion

This work simulated an 80 m high stack with an inner diameter of 3.5 m at a site of which basic atmospheric conditions were 293 K and 101325 Pa. The flue gas exhausting velocity from the bottom of the stack was fixed at 20.5 m/s, and the temperature of the flue gas was 388 K. The test wind velocities were 1 m/s and 5 m/s. Under these conditions, the plume dispersion and pollutant concentration distribution was compared numerically.

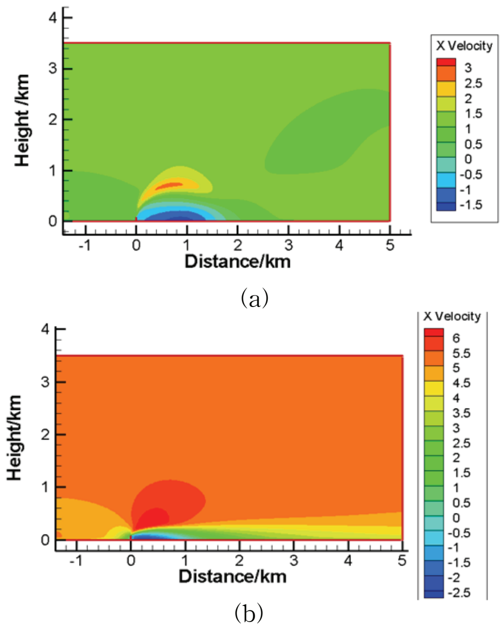



Fig. 2. Velocity contours at centre plane for stacks with different wind velocities: (a) 1 m/s; (b) 5 m/s.

Fig. 2 shows the velocity fields at the centerline plane as the model in different wind velocity conditions. The plume comes out from the stack is small, and then increases bigger and higher, finally reduces as distance further. Early stages of a kidney-shaped formation can be seen as the surrounding air is entrained in the plume and rises towards the top in the central region of the plume. From inside to outside, the flue gas velocity reduced layer by layer.

Comparing these two figures, both generate minus velocity (blue color) behind the stack. In addition, the smaller wind velocity is, the larger areas of minus velocity are. Large wind velocity makes the plume full, and it is better for flue gas and ambient air mixing, and diluting.

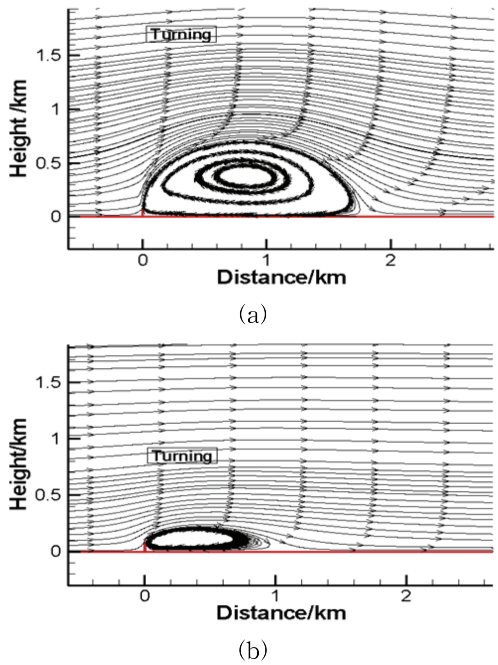
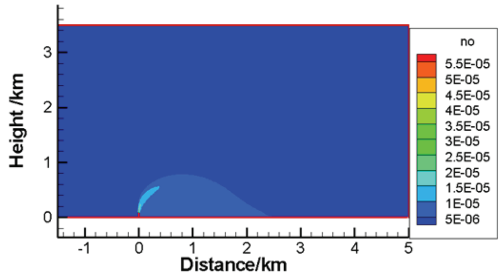



Fig. 3. Streamlines of plume subjected to different wind velocities: (a) 1 m/s; (b) 5 m/s.

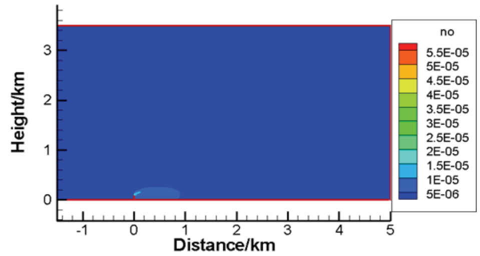
Fig. 3 presents the streamlines of plume in atmospheric cross flow. They are the partial enlarged drawings. In the combination of initial momentum and atmospheric turbulence, the flow rises with its temperature and velocity reduced. The maximum height of the plume appears at about 1550 m high 800 m away from the centre of the stack with 1 m/s wind velocity, and about 750 m high 700 m away from the centre of the stack with 5 m/s wind velocity. Low wind velocity, high plume rising height. After rising to the maximum height, the plume starts to descend and finally the plume keeps parallel with the ground.

In addition in Fig. 3, there is an eddy presents to the downstream behind the stack, which is the minus velocity area presented in Fig.2. Because the stack is set to wall boundary, the wind cannot pass by. In the downstream near the ground, stable air condition is affected by the upper atmospheric turbulence. Because of the large different wind velocities between the up and down, it is easy to generate an eddy which has a horizontal axis. The coherent vortex structure is the mechanism for downwash. It causes the plume force downward into the vertically oriented vortices being shed by the cylindrical stack, and the downwash material would be transported toward the ground [1].

Fig. 4 presents the gaseous NO diffusion profile. A certain scale of stagnant zone which could be resulted from the downwash corresponds to the flow pattern in the vicinity of the stack as shown in Fig. 3. However the concentration of NO has been diluted much, from the initial concentration 60 ppm to the ground-level concentration below 8 ppm, and it almost reduces by 90%. As the wind velocity increases, more air mixes with the flue gas and the dilute speed increases more efficiency. Therefore the concentration of NO pollutant is lower at the same position. Hence, large wind velocity is good for pollutant diffusion.




(a)




(b)

Fig. 4. NO Pollutant diffusion area with different wind velocities: (a) 1 m/s; (b) 5 m/s.

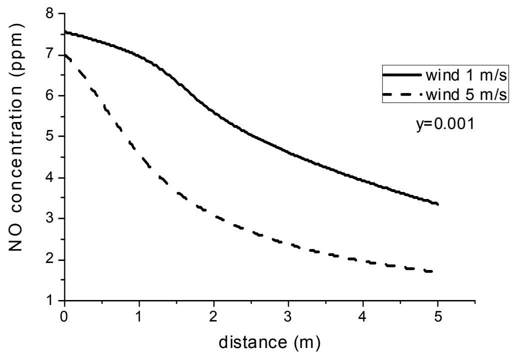



Fig. 5. NO Concentration curve at y=0.001 km with different wind velocities.

Fig. 5 presents the NO concentration graph at near ground plane (y = 0.001 km) with different wind velocities. The near ground plane is at humans breathing zone. Research about the pollutant concentration at these areas is necessary for people's health. These two curves have the same trends that NO concentration reduces alone with the increasing of the distance at y=0.001.

However, the reduced rate is large due to the large wind velocity.

From the Table 1, it shows NO Concentration distribution of the breathing zone according to the downstream distance at the distance at 3 km of the downstream, the concentration reduced to 4.6 ppm and 2.4 ppm, respectively, the wind velocity is 1 m/s and 5 m/s, while the attenuation ratio of the NO reach to 92.3% and 96%.

## 4. Conclusions

A two-dimension numerical simulation of a plume from an 80 m height stack in an atmospheric cross flow with different wind velocities is performed in this paper. From the analysis and comparison, it clarifies the airflow movement characteristic and the pollutant diffusion distribution. The conclusions are:

1) When the wind velocity is 1 m/s, the highest plume height presents to 1550 m high and 800 m far from the stack. It generates a big eddy in the downstream behind the stack. At the line of breathing zone, NO attenuation ratio reaches to 92%, the distance is 3 km far away the stack. And the pollutant distribution areas are larger than wind velocity 5 m/s.
2) When the wind velocity is 5 m/s, the highest plume height presents to 750 m high and 700 m far from the stack. The rise height is lower than wind velocity 1 m/s. It generates a small eddy in the downstream behind the stack. At the line of breathing zone, NO attenuation ratio reaches to 92%, the distance is 1 km far away the stack.
3) Wind velocity is one of the most important factors for flue gas diffusion. When it increases, the injection height is low, but it makes flue gas mix with air sufficiently, the flow field is more active, and the pollutant concentration reduces much more. Larger wind velocity makes less downwash. Therefore, large wind velocity is good for flue gas diffusion.

Table 1. NO Concentration Distribution with the Distance at Ground Level

| | | | Wind velocity 5 m/s | |
|---|---|---|---|---|
| X/ km | NO concentration/ ppm | Attenuation ratio/ % | NO concentration/ ppm | Attenuation ratio/ % |
| 0.1 | 7.6 | 87.4 | 7.0 | 88.4 |
| 0.5 | 7.3 | 87.8 | 5.8 | 90.3 |
| 0.8 | | 88.1 | 5.1 | 91.5 |
| 1.0 | 7.0 | 88.4 | 4.6 | 92.4 |
| 1.5 | 6.34 | 89.4 | 3.6 | 93.9 |
| 2.0 | 5.6 | 90.6 | 3. 1 | 94.9 |
| 2.5 | 5.0 | 91.6 | 2.7 | 95.5 |
| 3.0 | 4.6 | 92.3 | 2.4 | 96.1 |
| 3.5 | 4.2 | 92.9 | 2.1 | 96.4 |
| 4.0 | 3.9 | 93.5 | 1.6 | 96.7 |
| 4.5 | 3.6 | 93.9 | 1.8 | 96.9 |
| 5.0 | 3.3 | 94.4 | 1.7 | 97.1 |

## References

1. C. R. Johnston, D. J. Wilson, A vortex pair model for plume downwash into stack wakes. Atmospheric Environment 31(1), 13 (1996).
2. C. S. Konig, M. R. Mokhtarzadeh-Dehghan, Numerical study of buoyant plumes from a multi-flue chimney released into an atmospheric boundary layer. Atmospheric Environment 36, 3951 (2002).
3. D. Contini, P. Hayden, A. Robins, Concentration field and turbulent fluxes during the mixing of two buoyant plumes, Atmospheric Environment 40, 7842 (2006).
4. H. K. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics, The Finite Volume Method. Longman, New York(1995).
5. I. Y. H. Yau, Design of industrial chimneys. Asia Engineer 12,23 (1995).
6. J. Brown, D. F. Fletcher, CFD prediction of odour dispersion and plume visibility for alumina refinery calciner stacks, Process Safety and Environmental Protection, 83(3), 231 (2005).
7. L. Lloyd, G. David, S. Joseph, Development and evaluation of the PRIME plume rise and building downwash model. Journal of the Air and Waste Management Association 3,373 (2000).
8. R. E. Britter, Recent research on the dispersion of hazardous materials, industrial safety environment and climate programmer. Final Report of the European Commission Contract no.ENV4-CT95-4001, publication of the European Commission EUR 18198 EN(1998).
9. S. V. Patankar, D. B. Spalding, A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. International Journal of Heat and Mass Transfer 15, 1787(1972).
10. X. P. Zhou, J. K. Yang, R. M. Ochieng, Numerical investigation of a plume from a power generating solar chimney in an atmospheric cross flow. Atmospheric Research 91, 26(2009).