ATRIUM SMOKE FILLING PROCESS BY COMPUTATIONAL FLUID DYNAMICS

W. K. Chow and R. Yin
Department of Building Services Engineering,
The Hong Kong Polytechnic University, Hong Kong

ABSTRACT

Atrium buildings are commonly found in Hong Kong since 1980. Those spaces are usually crowded with people and so fire protection systems have to be installed for providing a safe environment. Smoke control system was identified to be important but no clear design guidelines are available because the smoke filling process was not well-understood.

In this paper, Computational Fluid Dynamics(CFD) or fire field model is applied to study the smoke filling pattern in atrium. Two common cases on smoke spreading out from a shop adjacent to the atrium; and with a fire located at the atrium floor itself were considered. Simulations with a modified form of the CFD package TEAM were performed. Application of the predicted results is illustrated.

INTRODUCTION

Smoke generated by fire in a shop would be carried by air and spread from the burning object to the other part of the building. This phenomenon is very important in providing fire safety in an atrium [1]. There are far too many atria constructed in Hong Kong and they are very different from those in elsewhere. It is very expensive to carry out full-scale burning tests for studying atrium smoke movement. Scale model studies are useful but scaling parameters have to be identified [e.g. 2, 3]. Apart from using physical modelling, Computational Fluid Dynamics (CFD) [4] or fire field model [5] is another possible way of studying the flow. This technique has been developed and applied for design purpose in the past two decades. This is believed to have good potential for achieving reliable results and is useful for predicting smoke movement in big enclosures although results have to be justified carefully [5]. There are many reasons why a CFD model is so attractive and the most attractive point is on the beautiful graphical presentation on the geometry, temperature fields and velocity vectors. The predicted 'microscopic' picture on the thermal environment described by the velocity vector diagram, the temperature, pressure and smoke concentration contours are useful for deriving the relevant macroscopic parameters for engineering purposes. The time required to get the predicted results is now highly shortened because of the development of efficient computing schemes and high speed computers.

Numerical modeling of smoke movement in an atrium was performed. Two common scenarios with smoke spreading for a shop to the atrium; and the fire in the atrium floor with a ceiling vent was studied. The CFD package TEAM [6] was selected as the fire simulator. Steady flow properties including the air velocity component, temperature, turbulence kinetic energy and its dissipation rate were predicted. The advantage of using this two-dimensional model is to have results predicted quickly even in a personal computer.

MODIFIED CFD PACKAGE TEAM

A set of time-averaged equations [4] for flow variables ϕ velocity components (u, v), temperature T, turbulence kinetic energy k, and turbulence energy dissipation rate ε given by the following form are solved:

$$\frac{\partial}{\partial x}(\rho u\phi) + \frac{\partial}{\partial y}(\rho v\phi) = \frac{\partial}{\partial x}(\Gamma_\phi \frac{\partial\phi}{\partial x}) + \frac{\partial}{\partial y}(\Gamma_\phi \frac{\partial\phi}{\partial y}) + S_\phi \tag{1}$$

where x and y are co-ordinate directions; Γ_ϕ and S_ϕ are the turbulent exchange coefficient and source term. Descriptions of ϕ, Γ_ϕ and S_ϕ are reported elsewhere are will be not reported in here.

The CFD code TEAM has the following characteristics [6]:

- The method of Centered Grid Nodes is used for defining the grid points and their associated control volumes with non-uniform mesh.
- The power-law differencing scheme (PLDS) was used for discretizing equation (1).
- A line-iterative method using Tri-diagonal Matrix Algorithm (TDMA) with alternating sweep directions was adopted to solve the discretized equations.
- The "Semi-Implicit Method For Pressure Linked Equations" (SIMPLE) is used to solve the set of velocity-pressure linked equations [4].
- Under-relaxation method is used for ensuring convergence.

NUMERICAL EXPERIMENTS

The first set of numerical experiment labelled as BAC1 is on simulating smoke spreading from fire in a shop to the atrium. The shop is of length 9 m and height 3 m, while the atrium is of size 10 m by 10 m. A fire of length 1 m and height 0.5 m is located at floor of the room with the door of 2 m high opened to the atrium as shown in Fig. 1. An opening of height 0.942 m was assigned from the atrium to the outside. Thermal power of fire was 300 kWm^{-1}. The geometry for BAC1 was divided into 62 by 72 parts along the x- and y-directions as shown in Fig. 1a.

The second set of numerical experiments is labelled as VENT1. The same atrium was considered but with a vent of width 2 m at the ceiling. The same 100 kWm^{-2} fire of size 1 m by 0.5 m was located at the center of the atrium floor. Only half of the atrium was simulated for VENT1 due to symmetry. The region was divided into 42 by 77 parts with the grid system shown in Fig. 2a. Note that the vent was not considered as a free boundary and computing region was extended to a height of 5 m above the roof.

Two kind of boundary conditions were used for the solid wall boundary and the entertainment boundary. Non-slip condition on the velocity components, thermally adiabatic wall, and "Wall-Function" method were used for solid wall boundary. For the free boundary, the pressure was prescribed with the derivatives of velocity components normal to the free surfaces set to zero. Ambient temperature was specified at points of in-flow and the derivative of the dependent variable was set to zero at the points of outflow. Boundary conditions (free boundary condition and solid boundary) for the two sets of simulations are shown in Figs. 1a and 2a.

Initial values of velocity and turbulence dissipation were assumed. Initial velocity fields were set to zero with initial temperature at 20°C, and initial pressure at 1.013×10^5 Pa. The initial turbulence energy k and turbulence dissipation ε were 0.25 m^2s^{-2} and 0.05 m^2s^{-3} respectively. The density of the fluid was taken as 1.293 kgm^{-3}, laminar viscosity was 1.8×10^{-5} Nsm^{-2}, the laminar Prandtl number was 0.7 and a constant specific heat C_p of 1020 $Jkg^{-1}K^{-1}$ was used.

CRITERIA FOR CONVERGENCE

The discretization form for the flow equation (1) is given by:

$$a_p\phi_p = \sum_i a_i\phi_i + b_i \tag{2}$$

Convergence criteria were in checking the coefficient a_p at a point P and its four neighbours a_i:

$$\frac{\sum|a_i|}{|a_p|} \leq 1 \quad \text{for all equations} \tag{3}$$

$$\frac{\sum|a_i|}{|a_p|} < 1 \quad \text{for at least one equation} \tag{4}$$

The sum of absolute residues $RESOR_\phi$ for air flow variables ϕ (other than pressure correction P') and the sum of absolute residues for the pressure equation (ϕ = p') $RESOR_m$ were calculated. Computations would be stopped where the maximum residues sum R_{max} of the variables ϕ (u, v, T, k, ε) and P' satisfying the following condition:

$$R_{max} = \text{Max}\ |RESOR_\phi\ ,\ RESOR_m| < R_{ref} \tag{5}$$

Value of R_{ref} was taken to be 5.0×10^{-4} for BAC1 and 1.0×10^{-3} for VENT1.

Negative values of k and ε, due to the strong heat source was prevented by following a scheme reported earlier [7].

RESULTS

The amount of heat transfer for BAC1 at the door of the shop (x = 9 m) and at the atrium entrance (x = 18 m) are shown in Table 1. Those results through the door ways (x = 0 m) and vents (y = 10 m) for VENT1 are shown in Table 2. It is observed that deviations of calculation are acceptable.

Velocity vectors for BAC1 is shown in Fig. 1b. Fresh air came in from the bottom of the entrance door into the shop through the atrium, heated up by the fire, then along the wall and to the ceiling of the shop. A small vortex were formed at the outside of the room soffit. It then flow out at the upper part of the atrium door. Under steady-stated condition, smoke moved only near the wall and ceiling of the atrium. No smoke would move to the atrium centre.

The velocity vectors of VENT2 is shown in Fig. 2b. Because of the vent at the ceiling of the atrium, fresh air would flow into the atrium, passes through the heat source, then went up to the atrium roof. There is a big vortex in the atrium and its ceiling.

Temperature contours for the two sets of numerical experiments are shown in Figs. 1a and 1c. For BAC1, upper part of atrium has the same air temperature as ambient. But for VENT1, temperature at the center of atrium is much higher than the ambient.

Vertical profiles of the turbulent viscosity in the atrium are shown in Figs. 3a and b. For BAC1, turbulent viscosity in the room is higher than it in the atrium. The turbulent viscosity is usually small near the wall, and has a peak between wall and ceiling. But in an atrium, there is a point where turbulent viscosity is equal to zero. This point is clearly illustrated in the vector diagram. For VENT1, turbulent viscosity is higher in the in-flow positions.

CONCLUSIONS

Two different scenarios for atrium smoke movement were simulated by the CFD package TEAM. Two sets of simulations were performed in the same atrium but with different arrangements. It is shown that two-dimensional simulations can give a reasonable prediction on the smoke movement pattern. More important, the CPU time would be less than 550 s in an Intel 486-DX. From the predicted velocity and temperature distributions, smoke filling process can be understood.

ACKNOWLEDGEMENT

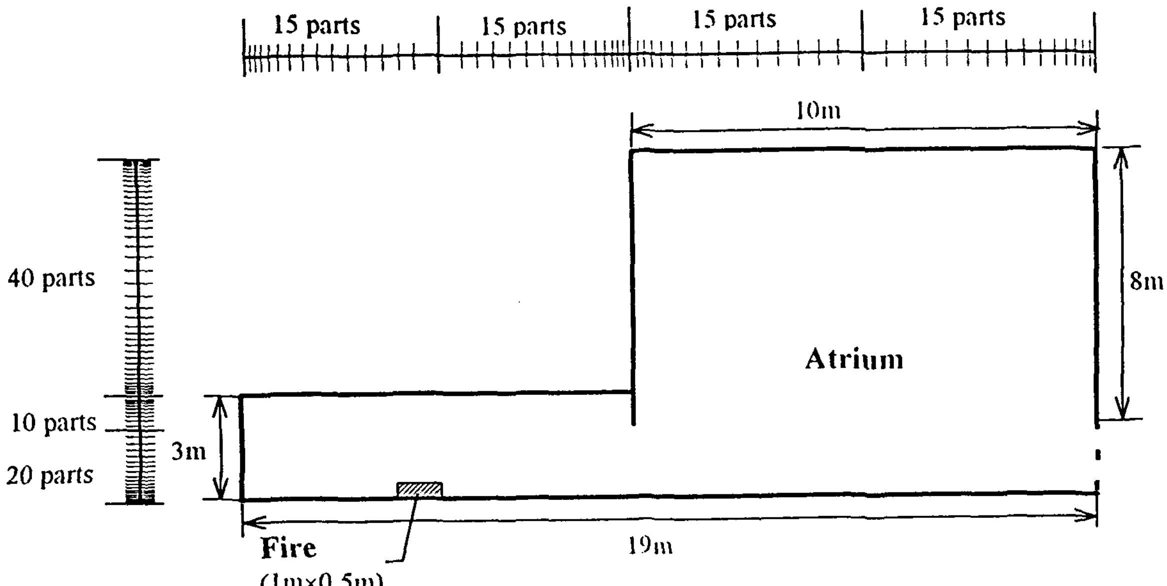
The project is funded (project number HKP23/94E, account number 357/001) by the Research Grants Council, Hong Kong. The authors wish to thank Professor Leschziner for allowing the use of the two-dimensional version of the program TEAM.

y/m	x=18.5m					x=9m				
	u/ms^{-1}	density/kgm^{-3}	T/°C	T-T$_0$/°C	Q/Wm^{-1}	u/ms^{-1}	density/kgm^{-3}	T/°C	T-T$_0$/°C	Q/Wm^{-1}
0.017	-1.42	1.23	34	14	-873	-1.85	1.293	20	0	0
0.054	-1.6	1.23	35	15	-1144	-1.82	1.293	20	0	0
0.094	-1.65	1.22	37	17	-1500	-1.77	1.293	20	0	0
0.139	-1.65	1.215	39	19	-1787	-1.71	1.293	20	0	0
0.188	-1.63	1.21	41	21	-2154	-1.64	1.293	20	0	0
0.241	-1.6	1.2	43	23	-2577	-1.56	1.293	20	0	0
0.3	-1.55	1.18	48	28	-3238	-1.46	1.293	20	0	0
0.365	-1.44	1.155	56	36	-4152	-1.36	1.293	20	0	0
0.437	-1.27	1.11	68	48	-5176	-1.23	1.293	20	0	0
0.515	-1.04	1.06	84	64	-5973	-1.09	1.293	20	0	0
0.602	-0.778	1.009	103	83	-6017	-0.921	1.293	21	1	-109
0.697	-0.487	0.9505	126	106	-5004	-0.729	1.293	21	1	-96
0.802	-0.164	0.892	152	132	-2146	-0.508	1.293	23	3	-219
0.917	0.193	0.831	183	163	3226	-0.25	1.293	27	7	-279
1.043	0.58	0.771	218	198	11951	0.05	1.293	42	22	191
1.182	0.987	0.715	256	236	24854	0.408	1.293	61	41	3221
1.336	1.4	0.668	294	274	42080	0.84	1.293	88	68	12128
1.504	1.8	0.632	326	306	62594	1.38	1.293	138	118	37798
1.689	2.17	0.6085	349	329	86094	1.91	1.293	249	229	111910
1.893	2.28	0.597	361	341	101464	1.71	1.293	287	267	128860
Total heat					290519					293406

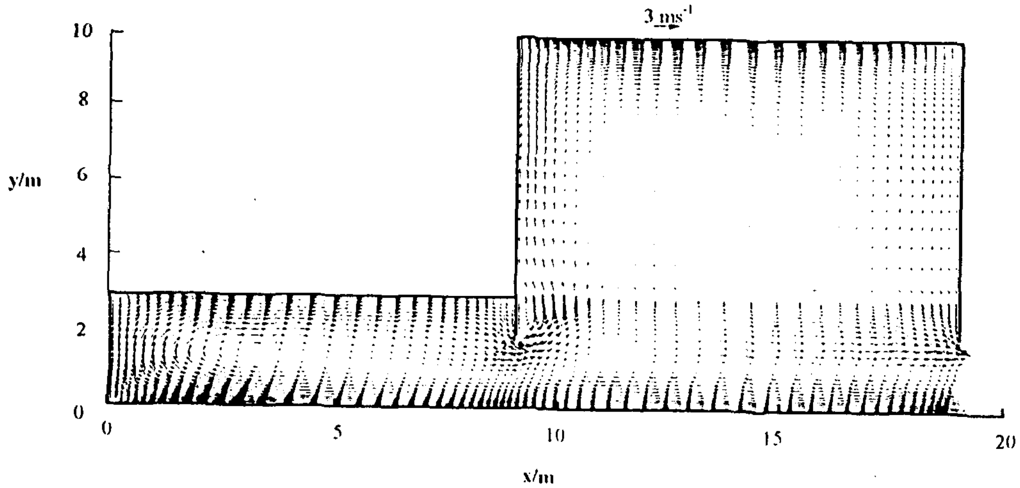
Table 1 Heat transfer at the doorway for BAC1
(Heat release rate of fire=300kWm^{-1})

y/m	Doorway x=0m			x/m	Vent y=10m					
	u/ms^{-1}	density/kgm^{-3}	M_{in}/kgm^{-1}s^{-1}		v/ms^{-1}	density/kgm^{-3}	M_{out}/kgm^{-1}s^{-1}	T/°C	T-T$_0$/°C	Q/Wm^{-1}
0.051	3.55	1.293	0.23	4.062	0.826	1.11	0.06	70	50	2899
0.107	1.94	1.293	0.14	4.125	1.020	1.10	0.07	71	51	3677
0.168	1.57	1.293	0.12	4.187	1.170	1.10	0.08	72	52	4232
0.236	1.37	1.293	0.12	4.25	1.300	1.10	0.09	73	53	4870
0.31	1.25	1.293	0.12	4.312	1.410	1.09	0.10	74	54	5248
0.392	1.16	1.293	0.12	4.375	1.500	1.09	0.10	74	54	5673
0.482	1.09	1.293	0.12	4.437	1.590	1.09	0.11	74	54	5918
0.581	1.04	1.293	0.13	4.5	1.660	1.09	0.11	75	55	6394
0.69	1.01	1.293	0.14	4.562	1.730	1.09	0.12	75	55	6558
0.81	1.01	1.293	0.16	4.625	1.780	1.09	0.13	75	55	6857
0.942	1.01	1.293	0.17	4.687	1.820	1.09	0.12	75	55	6900
0.942	(v)2.28	1.293	0.07	4.75	1.850	1.09	0.13	75	55	7126
				4.812	1.880	1.09	0.13	74	54	6997
				4.875	1.890	1.09	0.13	74	54	7148
				4.937	1.900	1.09	0.13	74	54	7072
				5	1.900	1.09	0.13	74	54	7186
ToTal			1.66				1.72			94762

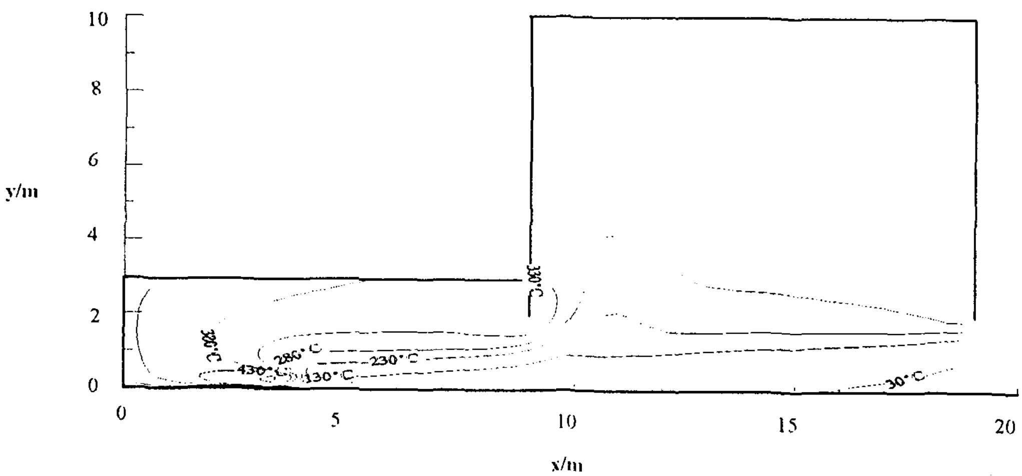
Table 2 Heat and mass transfer at doorway and vent for VENT1
(Heat release rate of fire=100kWm^{-1})



(a) Geometry

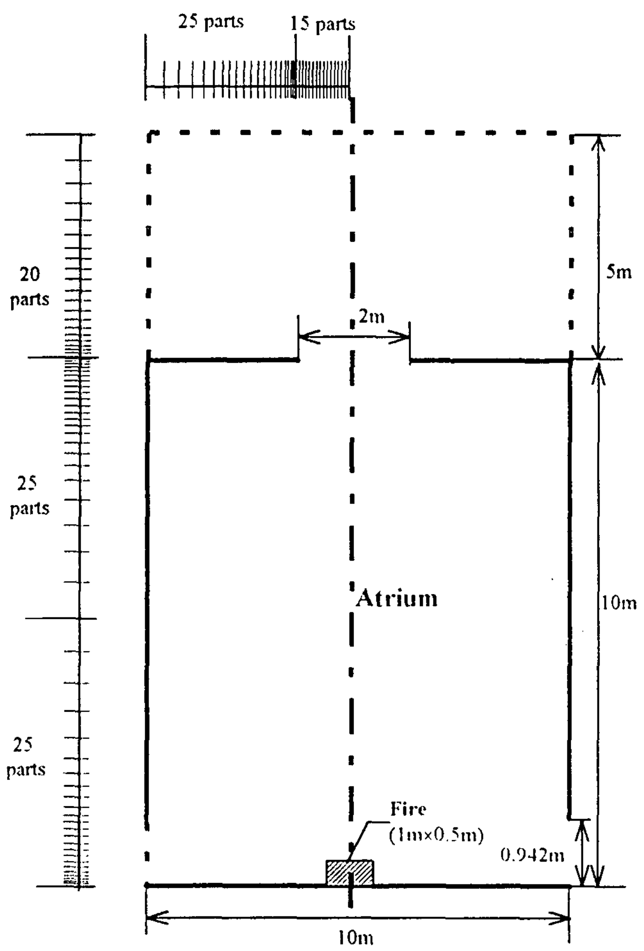


(b) Velocity vectors

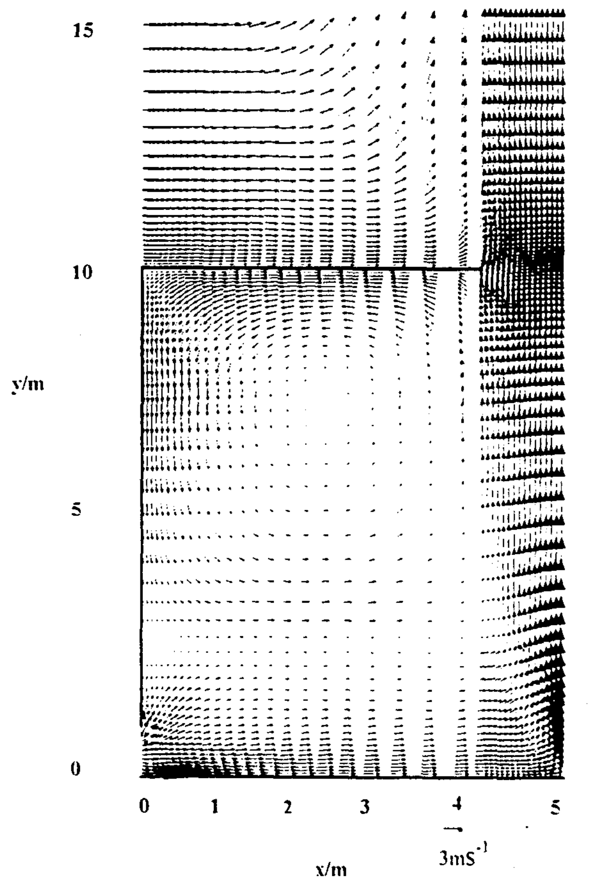


(c) Temperature contours

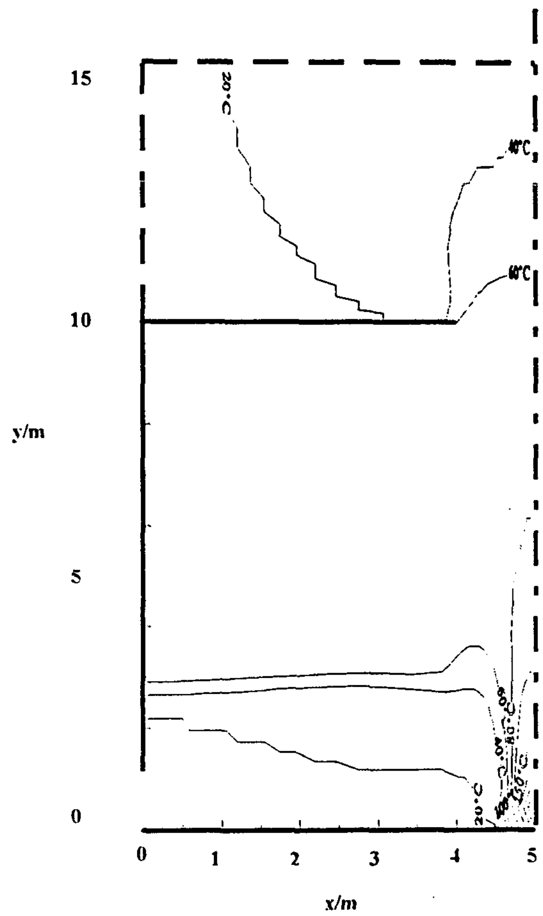
Fig. 1 Simulation BAC1
(Heat release rate=300kWm^{-1})



(a) Geometry



(b) Velocity vectors



(c) Temperature contours

Fig. 2 Simulation VENT1
(Heat release rate=100kWm^{-1})

REFERENCES

1. Chow, W.K. On the use of time constant for specifying the smoke filling process in atrium halls, Fire Safety Journal, 28(2), 165-177, 1997.

2. Quintiere, J.G. Scaling applications and fire research, Proceedings of International Symposium on Scale Modelling, 1988.

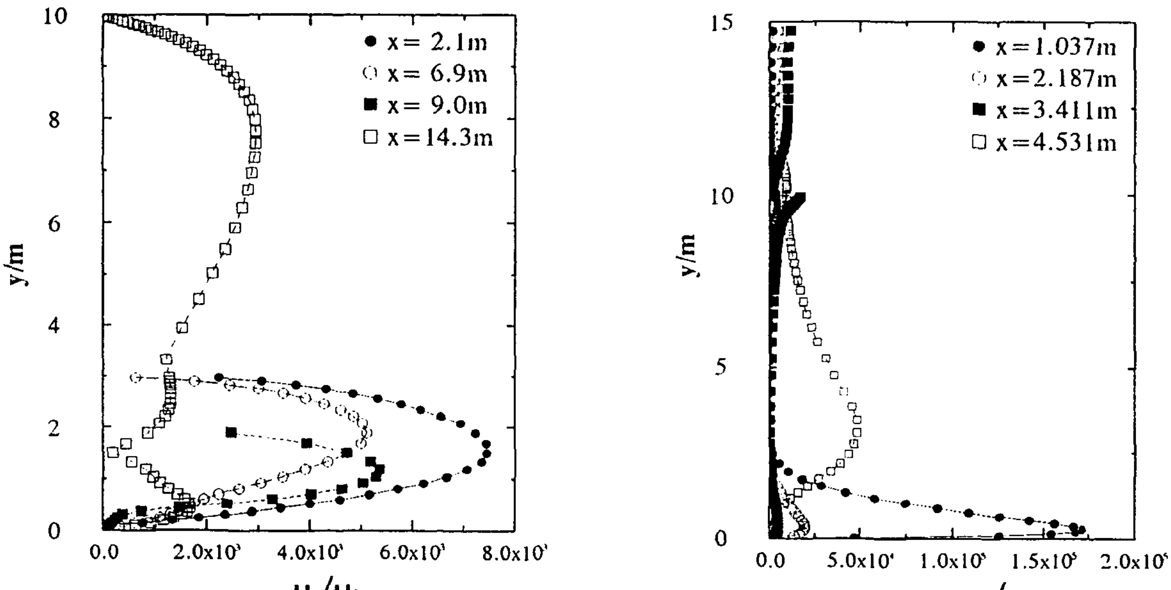
3. Nagaoka, T., Tsujimoto, M. and Okumiya, M. A smoke movement compared with real scale fire test (Kuramae Kokugikau) (in Japanese), Proceedings of the Annual Meeting of Architectural Institute of Japan, Part A, 1099-1100, 1991.

4. Spalding, D.B. Mathematical modelling of fluid mechanics, heat transfer and chemical reaction processes, Computational Fluid Dynamics Unit Report No. HTS/80/1, Imperial College of Science & Technology, 1980.

5. Chow, W.K. Use of Computational Fluid Dynamics for simulating enclosure fires, Journal of Fire Sciences, 13(4), 300-334, 1995.

6. Huang, P.G. and Leschziner, M.A. An introduction and guide to the computer code TEAM, Research Report, UMIST, Manchester, U.K., 1984.

7. Chow, W.K. and Leung, W.M. A short note on achieving comergent results in simulating building fire using the k-ε turbulent model, Numerical Heat Transfer, Part A, 17, 495-501, 1990.



(a) Simulation BAC1 **(b) Simulation VENT1**

Fig.3 Vertical varitions of turbulent viscosity