STUDIES ON THE VECTOR-FLOW CLEANROOM: NUMERICAL SIMULATION AND EXPERIMENTS

Guangbei Tu Wenhao Chen Lai Wang
Tianjin University P.R.China

ABSTRACT

Based on K-ε two equation turbulence model, we used PHOENICS 1.4 and numerically simulated air distribution and contamination field under different conditions in a vector-flow clean room. Special mesh system was introduced to deal with the quarter-circle-shaped inlets. Model experiments were also made. By analysis of numerical as well as experimental results, we made some predictions about flow characteristics, contaminant control effect and ventilation performance of this energy-saving clean room.

INSTRUCTIONS

Cleanrooms have been more and more widely applied in Chinese modern industrial and scientific fields in recent years. Traditionally, they are mainly divided into two classes – laminar flow and turbulent flow. For most engineers, laminar flow cleanroom is always the first choice for a clean space with a cleanliness class of class 100. However, its costs and energy consumption are relatively high, which, to some extent, limits its application in China. So how can we meet the required cleanliness class at the cost of minimum cleaned air? Many efforts are devoted to find out some new suitable flow patterns. As a result, a new type of cleanroom – vector-flow cleanroom has been in research and application in China.

When quarter-circle-shaped inlets are installed at one high corner and return outlets are installed at the opposite low corner, the airflow will be almost one-directional and it can push the contaminants out quickly. We call this flow pattern “vector-flow”. In this paper, a simplified 2-D model and a more complicated 3-D model are established and analyzed mainly by numerical methods. Field experiments are also made to get a more comprehensive understanding about its contaminant control capacity.

PHYSICAL MODEL

If both inlets and outlets are fully distributed along walls, the flow can be approximately considered as two-dimensional flow in plane of X-Z and thus a simplified 2-D physical model is created, as showed in Figure 1. However, this doesn’t always appear in true cases. In most projects, rectangular return grilles are used, so we further established a 3-D model which has the same size as the real model , showed as Figure 2. The size of each outlet is 500mm×400mm.

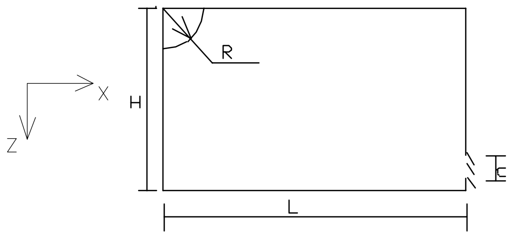


Figure 1 2-D Model

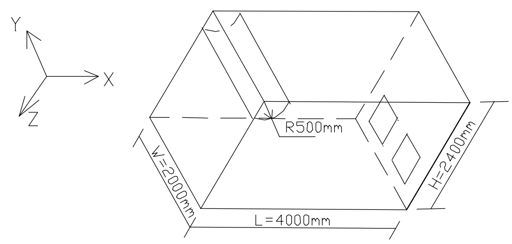


Figure 2 3-D Model

GOVERNING EQUATIONS

The turbulence model used in this study is K-ε two equation model, which has successfully applied in analyzing airflow pattern in many other cleanrooms.

In order to simplify the problem, the following assumptions are made:

(1) Values of the whole temperature field in the vector-flow cleanroom are uniform and the flow is isothermal.
(2) Only stable status is considered. That is, all variables are only functions of spatial coordinates.
(3) Particles with diameter less than 0.5um have a good following ability and their velocities are the same as the airflow.
(4) The contaminant source is considered to be a point source, namely, it doesn't have any physical volume.

Meantime, physical quantities are made dimensionless by representative quantities. These quantities are the radium of inlet R, its radial mean velocity UR and the ratio of generation rate of contaminants Q to supply air volume V (C0=Q/V).

Under above conditions, the dimensionless governing equations in the general elliptic form can be written as:

$$\partial(U_j\phi)\Big/\partial x_j - \partial(\Gamma_t \cdot \frac{\partial\phi}{\partial x_j})\Big/\partial x_j = S_\phi \quad (1)$$

The parameters ϕ, Γ_ϕ and S_ϕ for each equation are listed as follows:

ϕ	Γ_ϕ	S_ϕ
1	0	0 (Continuity)
U_j	$\gamma+\gamma_t$	$-\partial P/\partial x_j$
K	$1/\mathrm{Re}+\gamma_t/\sigma_k$	$\gamma_t(\partial U_i/\partial x_j+\partial U_j/\partial x_i)\partial U_i/\partial x_j-\varepsilon$
ε	$1/\mathrm{Re}+\gamma_t/\sigma_\varepsilon$	$C_1C_uK(\partial U_i/\partial x_j+\partial U_j/\partial x_i)\partial U_i/\partial x_j - C_2\varepsilon^2/K$
C	$1/\mathrm{Re}+\gamma_t/\sigma_c$	QR/U_RC_o

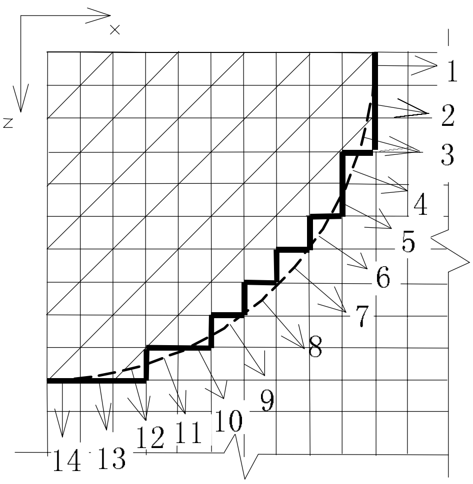
Empirical coefficients are:

$C_\mu=0.09, C_1=1.44, C_2=1.92, \sigma_k=1.0, \sigma_\varepsilon=1.3, \sigma_c=1.0$

SIMULATION METHOD

A CFD code PHOENICS is adopted to simulate this problem. It is based on finite volume approach and SIMPLEST algorithm for solving Navier-Stokes equations and conservation equations for heat and mass transfer. A staggered grid is used in PHOENICS. The solution procedure for variables, the difference scheme and many other parameters should be carefully selected by respective users.

Some considerations were taken into when we developed our user program include:



1. $U=U_R$ $V=0$ $W=0$
2. $U=U_R\cos9^0$ $V=U_R\sin9^0$ $W=0$
3. $U=U_R\cos15^0$ $V=U_R\sin15^0$ $W=0$
4. $U=U_R\cos21^0$ $V=U_R\sin21^0$ $W=0$
5. $U=U_R\cos27^0$ $V=U_R\sin27^0$ $W=0$
6. $U=U_R\cos34^0$ $V=U_R\sin34^0$ $W=0$
7. $U=U_R\cos41^0$ $V=U_R\sin41^0$ $W=0$
8. $U=U_R\cos49^0$ $V=U_R\sin49^0$ $W=0$
9. $U=U_R\cos56^0$ $V=U_R\sin56^0$ $W=0$
10. $U=U_R\cos63^0$ $V=U_R\sin63^0$ $W=0$
11. $U=U_R\cos69^0$ $V=U_R\sin69^0$ $W=0$
12. $U=U_R\cos75^0$ $V=U_R\sin75^0$ $W=0$
13. $U=U_R\cos81^0$ $V=U_R\sin81^0$ $W=0$
14. $U=0$ $V=U_R$ $W=0$

Figure 3 Mesh system and velocity calculation at the inlets

(1) Inlet Boundary Conditions

The true quarter-circle shape is replaced by small steps. We assume the radial velocity is uniform with a value of UR = 0.7m/s and the specific component velocities in each grid can thus be calculated, showed as Figure 3. Values of other variables are specified as k = 0.04, ε = 0.008, C = 0.

(2) Exit Boundary Conditions

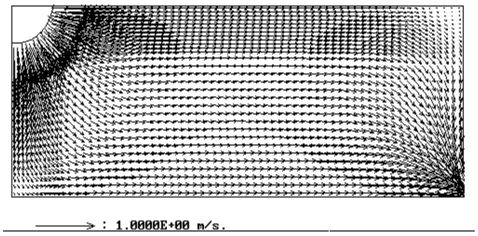
Only mass outflow conditions are needed in PHOENICS. They can be represented by a high-coefficient pressure boundary condition.

(3) Wall Boundary Conditions

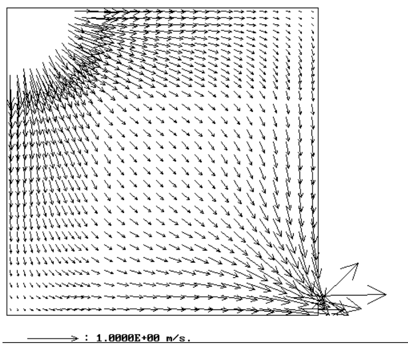
Empirical wall functions are used in conjunction with the high-Reynolds-number K-ε model.

To acquire the converged solution, linear under-relaxation is used for P and false-time-step under-relaxation is used for U, V, W, k and ε. After 1500 iterations, the whole-field residues of all variables are less than 10 –6 for 2-D model and less than 10 –4 for 3-D model.

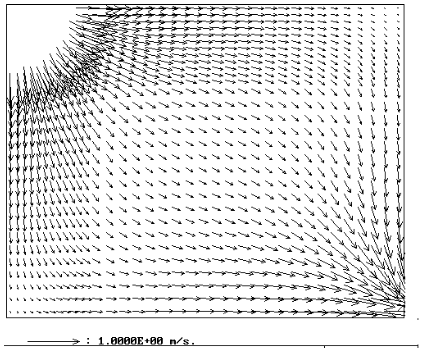
CALCULATING RESULTS AND DISCUSSIONS



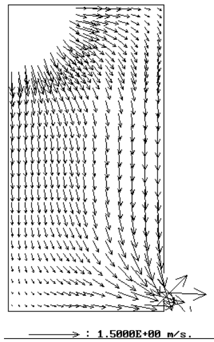
1) L*H = 6m*2.4m ; H/L = 0.4



2) L*H = 3m*2.4m ; H/L = 0.8



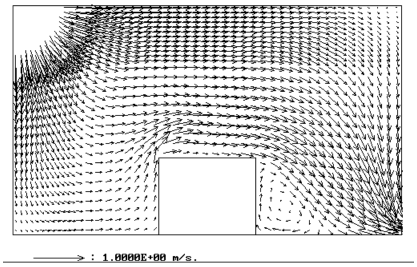
3) L*H = 2.4m*2.4m ; H/L = 1.0



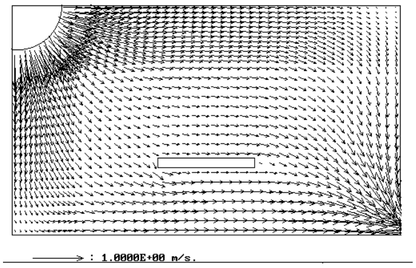
4) L*H = 1.2m*2.4m ; H/L = 2.0

Figure 4 Velocity vectors for 2-D cases with different H/L (R=0.5m, c=0.2m)

Figure 4 illustrates velocity vectors for 2-D cases with different H/L. The flow pattern shows obvious difference with the variation of H/L. When H/L equals to 1.0, the angle between airflow and horizontal direction is approximately 450. This angle is reduced with the decrease of H/L and the flow pattern is more and more close to horizontal laminar flow. On the other hand, when H/L is increased, the flow pattern is gradually close to vertical laminar flow. For all cases, small vortexes exist only at the left-low corner and the right-high corner. The flow pattern in working areas is similar to laminar flow, but the velocity uniformity is worse than that of laminar flow.



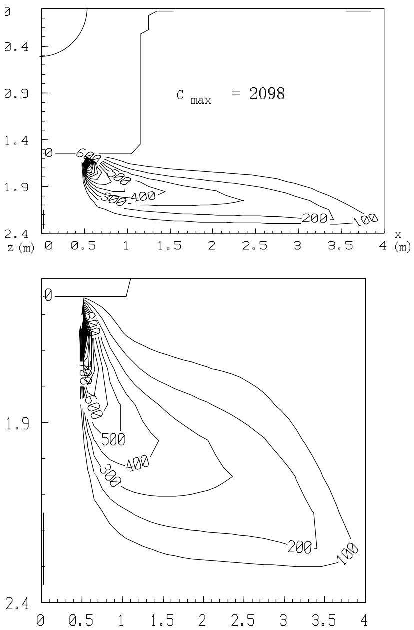
a) working bench 1.0m*0.8m (box)



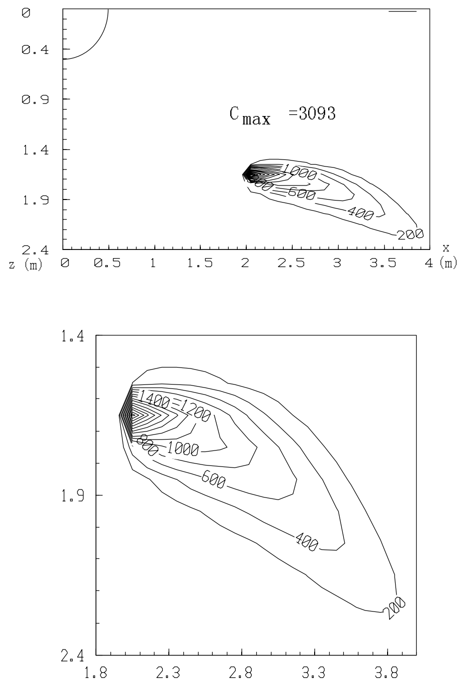
a) b) working bench 1.0m*0.1m (plane)

Figure 5　Velocity vectors for 2-D case with working bench　(L*H=4m*2.4m, R=0.5m, c=0.2m)

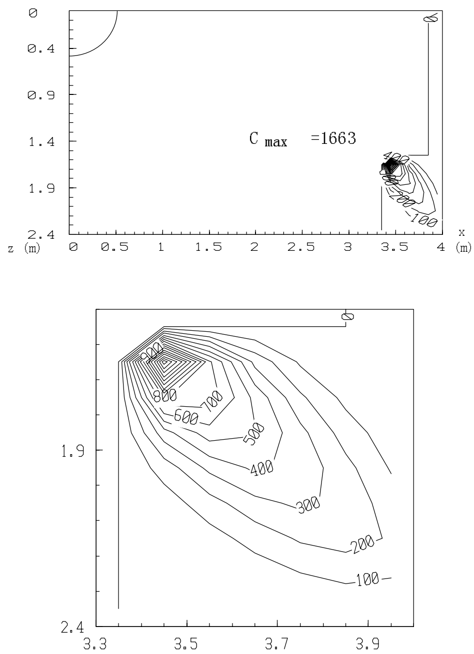
Figure 5 presents velocity vectors for 2-D case with a working bench (plane/box). The flow pattern is somewhat changed due to the hindrance of box (or plane). Vortex appears in the area behind the box. So the best place for the working bench is in the middle or back of the cleanroom.



(a) point source at location 1 (x=0.4m, z=1.6m)



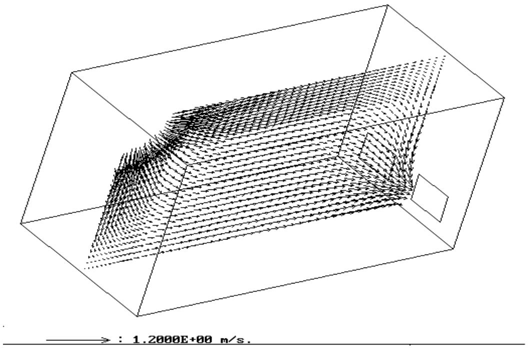
(b) point source at location 2 (x=2m, z=1.6m)



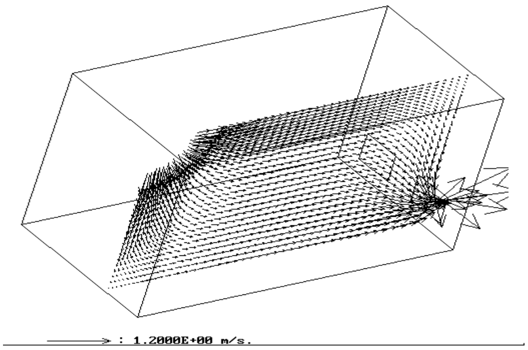
(c) point source at location 3 (x=3.5m, z=1.6m)

Figure 6　Contaminant contours for 2-D case with a point source (L*H=4m*2.4m, R=0.5m, c=0.2m)

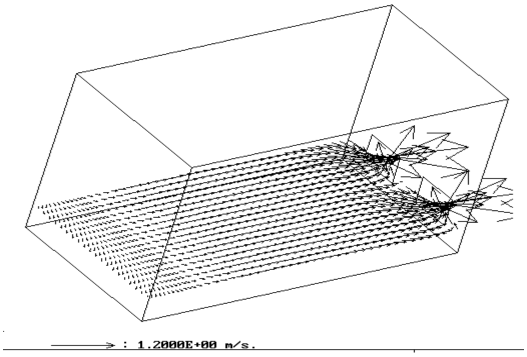
Figure 6 shows the contaminant distribution with a same contaminant source (generation rate Q=10000 Particles / (L • s)) at 3 different locations for a 2-D case. There is a close relationship between the location of the source and its diffusion areas: the nearer it is to the outlets, the smaller the contaminated areas are. Generally speaking, the contaminant dispersion is restrained by one-directional flow and contaminants mainly go directly towards the outlets. Its contamination control effect is obviously better than that of turbulent flow.



a) Vertical section y/W=0.5



b)Vertical section y/W=0.3



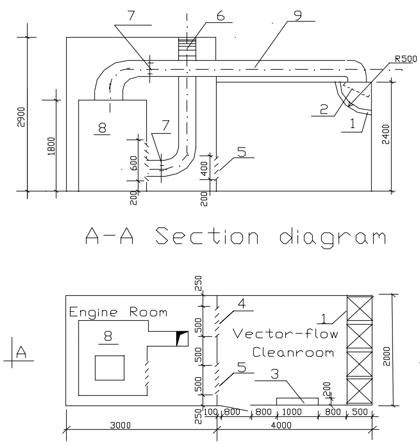
c)Horizontal section z/H=5/6

Figure 7 Velocity vectors of different sections for 3-D model

Figure 7 presents velocity vectors in different sections for the 3-D model. There is only slight difference between different vertical sections due to the change of shape and location of the outlets. The flow pattern is similar to that of 2-D case.. So we can predict that the effects of outlets on the flow pattern is relatively small and the simplified 2-D model is reasonable.

FIELD EXPERIMENTS

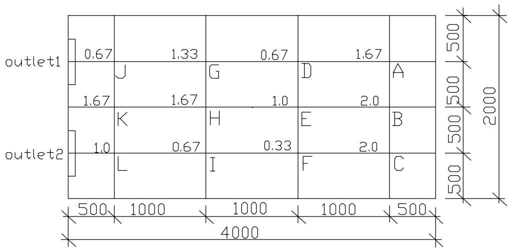
The contaminant concentration in a cleanroom is closely related with its air distribution. A good flow pattern should make the contaminants brought out quickly to assure a high cleanliness class in the whole working area. So we measured the airborne particles level (≤0.5um) in a model cleanroom, whose size is 4m*2m*2.4m. The experimental system is showed as Figure 8.



1.high-efficiency vector-flow inlets
2.rectifier 3.passing box
4.return outlet1 5.return outlet2
6.fresh-air duct 7.valves
8.air-cleaning unit 9.supply-air duct

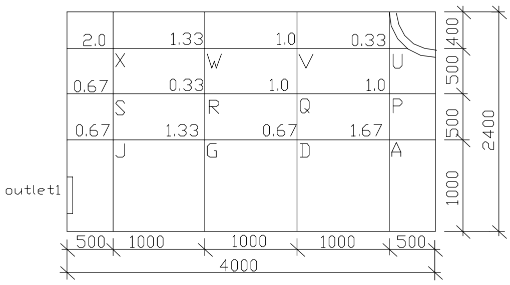
Figure 8 Experimental system

The room air exchange number is 148 per hour. The counters' sampling rate is 1ft3/min and the sampling time is set at 1 min. For each sampling point, no less than 3 samples are taken and their mean value is calculated. The distribution of sampling points and statistic results are showed as Figure 9.



$L = 12 \quad \overline{N} = 1.2 / ft3 \quad \sigma = 0.55$

(a) 1.0m Working Plane



$L = 12 \quad \overline{N} = 1.0 / ft3 \quad \sigma = 0.49$

(b) Section including return outlet1

Figure 9 Particle Count Results (≥0.5um)

Where, $\overline{N} = \frac{\sum_{i=1}^{L} Ai}{L}$ –Average concentration at the selected section

（Ai—Average concentration at ith sampling point; L—Total number of sampling points at the selected section ）

$$\sigma = \sqrt{\frac{\sum_{i=1}^{L}(Ai - \overline{N})^2}{L-1}}$$ – Standard Deviation

The experimental results suggest that its contaminant control effect is almost as good as that of laminar flow, which is correspondent with simulation results.

CONCLUSIONS

Although the numerical and experimental results can't be compared directly, almost the same conclusions can be drawn:

(1) When inlets and outlets are fully distributed along walls, the vector-flow cleanroom has a good flow pattern and contaminant control effect, similar to the laminar flow. It can meet a high cleanliness such as class 100 with a relative low airflow rate.

(2) The air distribution in a vector-flow clean room is mainly determined by inlets and the effect of outlets is relatively small. So a simplified 2-D model can be used to replace the actual 3-D model for the purpose of designing more quickly and easily.

ACKNOWLEDGMENTS

The author would like to express appreciation for the help of China Cleaning Engineering Company in Tianjin. They provided the model clean room for field experiments.

NOMENCLATURE

U_j = components of velocity vector
P = mean pressure
C = contaminant concentration
UR = radial velocity at the inlets
k = turbulence kinetic energy
ε = turbulence dissipation rate
γ = molecular kinetic viscosity
γt = eddy kinetic viscosity
Re = Reynold Number

Guangbei Tu, Prof. Dep. Civil Eng.
Fax / Tel: 86-022-27404510
E-mail: gbtuwala@public.tpt.tj.cn