Application of Porosity in Calculations of the Wind-Induced Pressure Distribution around a Model Cabin

H DONG and **W FAN**
State Key Laboratory of Fire Science, University of Science and Technology of China
Hefei, Anhui 230026, China

ABSTRACT: During the process of ship fire, the influence of wind is very important. The wind flow sets up pressure variations around the cabin , in which the flow velocity and direction play an important role in determining the smoke movement within the cabin and the rate of ingress of external smoke. The calculations have been performed using Porosity *concept and method. Data from experiments have been compared with those from the simulations. The calculation results are found to be in good qualitative agreement with the experimental data.*

Key Word: Porosity, Pressure distribution, Numerical simulation.

1 INTRODUCTION

In the event of ship fire, the fire spread and the smoke movement are affected by factors such as the cabin structure, the fuel properties and the environment conditions, in which, the influence of the air flow is very important. It is well known that the air flow is partially determined by both the prevailing wind condition and the ship movement state. The pressure distribution around the cabin surface will be changed with the change of the speed and direction of the air flow as well as the ship movement. The fire process will be affected by the air being driven in on windward faces and smoke being sucked out on downwind faces, and the wind-induced pressure distribution would determined how quickly the air would be driven in and the smoke would be sucked out the fire cabin .So in the ship fire research, it is necessary to determine the pressure distribution with reasonable accuracy and at low cost. This work is also relevant to determination of the survivable and escaping time of the crew in the cabin as well as the ship.

Experimental studies of the influence of the wind -induced pressure field have been performed by Kandola[1]. The results showed that a proper allowance for the effect of the wind must be made if the smoke ingress rates is to be well predicted.

Numerical simulation is another available means which can be used to calculate the wind-induced pressure distribution. In the early time, the BFC(Boundary Fitted Coordinate) and the source terms added are usually used to calculate the flow field when any object exists in flow field,. and in these methods the process of mesh is very complex, and the calculation accuracy is not well also. McCaughey and others have used the hydrodynamic method calculation model FLOW3D to simulate pressure distribution on the surface of a experiment model. But in their simulation, the K-ε equation was replaced by the experience formula [3],which made the reliability of the method was limited . Moreover, it is too complex to process for the calculation regions were cut apart to five blocks.

In order to make the pressure distribution calculation easier, the mathematical physical model was built up in this paper according to the determined wind velocity and direction. The pressure distribution on the surface of the model cabin was simulated using the porosity concept and method.

2 THE DESCRIPTION OF POROSITY CONCEPT AND METHOD

Porosity is defined as the ratio of the sizes of an object in special regions with the sizes of the regions. Generally, the flow field will be distorted if there is an object in the flow field. At the very first, the "porosity" concept was used to process the change of flow field when there are some objects exist in the flow domain [2]. In order to simplify the calculation, the concept "porosity" was introduced and used to calculate the pressure distribution on the model cabin surface in this paper.

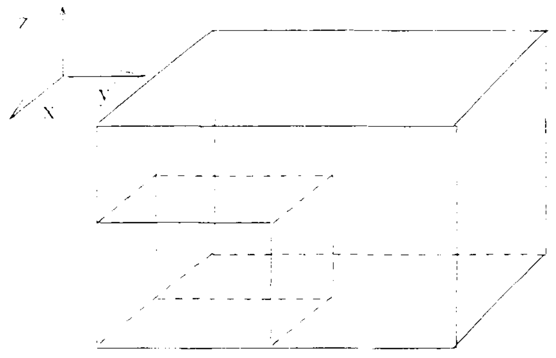


Fig. 1 Sketch map of porosity concept

In the control volume analysis, the active faces and volume of the convective transport and diffusive transport in the discrete equation is the faces and volume of the control volume. When there is any object in the numerical domain, as shown in Fig.1, the active faces and volume are different with that of the control volume. The active faces A_i' and volume V_i' can be expressed to multiply factors as follows:

$$\begin{cases} A_i' = A_i P_{fi} \\ V_i' = V_i P_{vi} \end{cases} \qquad (1)$$

Where P_{fi} is the face porosity and P_{vi} is the volume porosity.

For the non-uniform crisscross meshes were used, it should be careful when the porosity value in the velocity meshes is determined . The concerned porosity values should be determined according to the ratio between the mesh sizes and the sizes taken by the cabin in the mesh

3 DESCRIPTION OF THE CALCULATION MODEL

It is clear that the flow in this case is very different with the high speed flow. It is necessary to consider the cabin surface layer in the wind field and study how to used a simple numerical method to build up a general calculation model. The calculation in this paper refereed the basic condition of the Kandola experiments, and aimed to verified the numerical method.

As noted before, the problem can be solved by defining the steady-state, environment temperature, incompressible and turbulent flow around the cabin. The K-ε equations and Log-law wall functions were used to simulate the turbulent flow. There are many successful example for the K-ε method used in engineering, furthermore we can not get enough data to ensure the reasonable application of more complex model such as Reynolds stress model. Accordingly, the governing equations are made up with continuity equation, momentum equation, turbulent kinetic energy equation and turbulent dissipation equation, which can be written in a general form as follows,

$$\frac{\partial(\rho\Phi)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j\Phi)-\frac{\partial}{\partial x_j}\left(\Gamma_\Phi\frac{\partial\Phi}{\partial x_j}\right)=S_\Phi \tag{2}$$

Where Φ stands for the dependent variables, Γ_Φ for exchange coefficient, and S_Φ for source term in general equation. They are listed in Tab. 1:

Tab. 1 the meanings of variable Φ, exchange coefficient Γ_Φ, and source term S_Φ in general equation

equation	Φ	Γ_Φ	S_Φ
mass continuity	1	0	0
x-momentum	u	μ	$-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial y}\right)$
y-momentum	v	μ	$-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial y}\right)$
z-momentum	w	μ	$-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial z}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu\frac{\partial w}{\partial z}\right)-\rho g$
turbulence kinetic energy	k	μ_t/σ_k	$G-\rho\varepsilon$
turbulence dissipation	ε	μ_t/σ_ε	$\frac{\varepsilon}{k}(C_1-C_2\rho\varepsilon)$

In Tab. 1, μ represents an effective viscosity which is the sum of the turbulent viscosity μ_t and the laminar viscosity μ_l, that is: $\mu=\mu_l+\mu_t$, turbulent viscosity can be get with the formula:

$$\mu_t=C_\mu\rho k^2/\varepsilon。$$

G represents the turbulent kinetic energy production term, which is defined as:

$$G=\mu_t\left(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial u_i}\right)\frac{\partial u_i}{\partial u_j}-\frac{\mu_t g_i}{\sigma_t T}\frac{\partial T}{\partial x_i}$$

$p=p_0+\rho gz$, ρ denotes the air density;p and $p_0$denotes the pressure at the given location and the reference pressure far from the cabin respectively .g is the acceleration of gravity, where the constants are : $C_\mu=0.09$, $C_1=1.44$, $C_2=1.92$, $\sigma_k=1.0$, $\sigma_\varepsilon=1.3$.

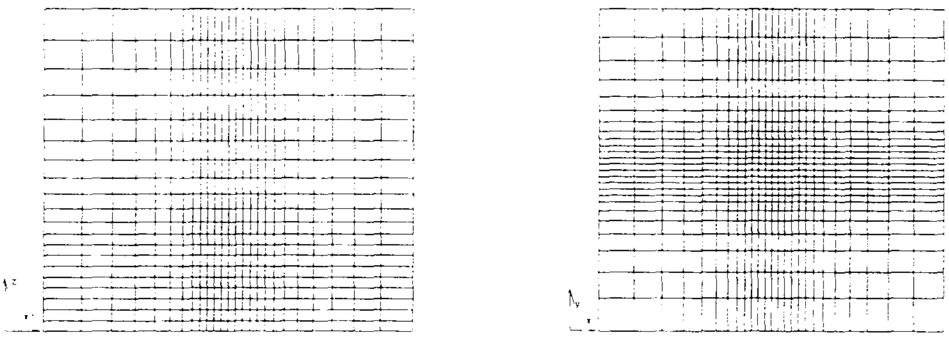


Fig. 2 the velocity meshes used in the numerical simulation

The simulated calculation model is a simple cube, which is placed in the bottom of a 4(m) ×(5m)×3(m) calculation field center. The geometry of the cube are 0.4(m)×0.4(m)×0.4(m). The meshes used in the numerical simulation are non-uniformed grids 20×30×20. In order to get enough meshes number on the surfaces around the cabin, two kinds of mesh size ratio were used in the flow field , that is using little scale meshes in the cabin region and larger scale meshes in else region . The meshes which were taken by the model is (10~17)×(10~17)×(1~9). Calculation used the SIMPLEC method. Iteration of the equations was continued with the main-line scanning until the total error in the equation was less than 0^{-4}, Under-relaxation factor is 0.4.

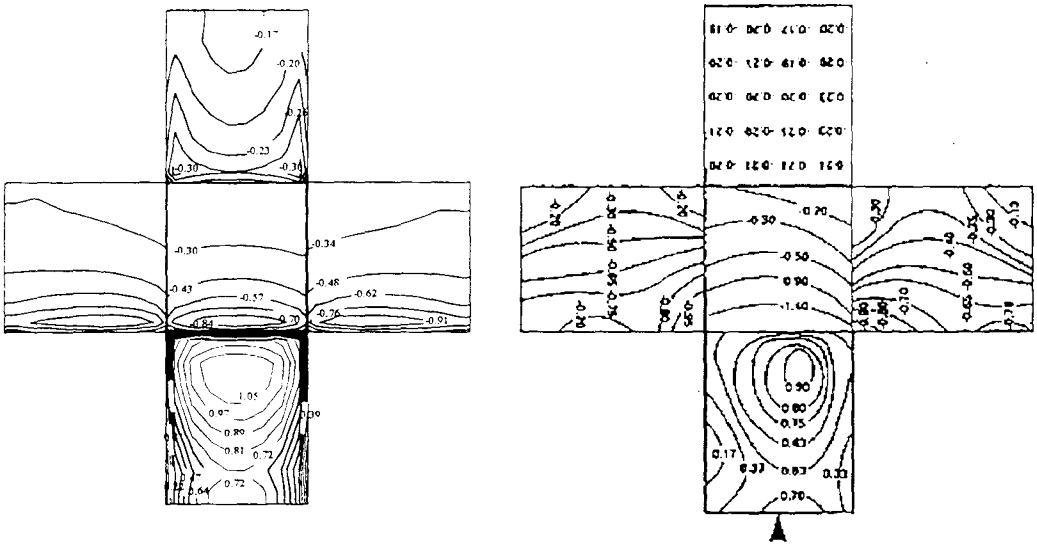
The velocity at the inlet was specified using the date given in equation(3)

$$V/V_{max} = (Z/Z_{max})^{0.32} \qquad (3)$$

where is horizontal velocity and Z is the vertical coordinate. According to the Kandola experiment, which is a full turbulence flow tunnel test, we use the same constants Z_{max}=1(m) and ${}_{max}$=10(m/s). The bottom and the two other wall are defined as no-slip boundary, that is V=0. The outlet denotes environment condition. The full-porosity method was used to process the region taken by the model, that is if a mesh is taken by the model ,the porosity value will be zero. The velocity source terms should be changed correspondingly in the dispersed equation.

4 CALCULATION RESULTS

The simulating calculation results were shown in Fig. 3 (a). For easy to compare, we use a non-dimensional quantity, the pressure coefficient which is defined as : $C_p = \frac{(p - p_0)}{\frac{1}{2}\rho V^2}$, where V is free stream velocity. The pressure coefficient distribution on the up face of the model was shown in the center of Fig. 3(a), up and down showed the pressure coefficient distributions on the front and rear faces. Left and right shown the pressure coefficient distribution on the two flank. Comparing to the Kandola experiment results, it showed that the both pressure coefficient distributions are very similar. It is clear that the pressure is positive on the front face and negative over the other face. The pressure distribution can easily be explained by the fact that the flow separates around the body and that there is a wake region downstream of it . It is worth noting that the pressure distribution in Kandola experiment data was not symmetric although the incident airflow was normal to the front face of the block. This is caused by asymmetries in the apparatus or the approaching flow . The computational results shown in Fig 3 (a) give a symmetric results . It shows that the numerical simulation is the best way to removed any asymmetries .By the way there are vortex in the rear field of the model which is clearly shown in the computational results , So the case ,actually, is a quasi steady state process.



(a) (b)

Fig. 3 . Comparison of experimental and calculated results

5 DISCUSSION AND CONCLUSIONS

Comparing to the experiment data, the calculating results are found to be in good qualitative agreement with the experiment results. It shows that the model ,with the improvement in this paper using the porosity concept, can be used to predicate the pressure distribution around a cabin calculation in different wind conditions. The validation work presented in this paper provides valuable base for the fire and smoke spread prediction in ship cabin fire . Because there are vents such as windows and doors , the cabin can not be looked as a solid and the model application will be more complex ,so more detail consideration is needed.

REFERENCES

1.Kandola,B.S.,A wind tunnel building model for the investigation of smoke movement problems. Fire Safety Journal, 1986,10:203-209.

2 . Fan Weicheng, Porosity and its application in calculation hydrodynamics, Journal of China University of Science and Technology, 1985, 15(4)

3. McCaughey M. &Fletcher D.F., Calculations of the Wind-induced Pressure Distribution on a Model Building ,Fire safety journal 1993, 21: 189-205

4. Soonil Nam and Robert G.Bill, Jr, Numerical simulation of Thermal Plumes, Fire Safety Journal 21 (1993) 231-256.