MARKET OPPORTUNITIES FOR ADVANCED VENTILATION TECHNOLOGY
22ND ANNUAL AIVC CONFERENCE
BATH, UNITED KINGDOM, 11-14 SEPTEMBER 2001

Full representation of supply openings for indoor airflow simulation

Bin ZHAO, Xitanting LI, Qisen YAN
Department of Building Science, Tsinghua University, Beijing, China

Abstract  The paper reviewed the supply opening models of describing inlet boundary conditions for indoor airflow simulation firstly. Then examples of isothermal free air jets from a grille are presented to validate full representation of supply openings. Some results by the simplified method called N-point supply opening model, are also showed to compare with the full representation method. Four different outlet conditions of a grille are studied. The $k-\epsilon$ turbulence model is applied. Comparison between simulated results and measured data shows that full representation is a reliable method for inflow boundary conditions description. Therefore, some other new models or methods can be developed based on full representation method as it is a universal and trusty method for describing inlet boundary conditions of indoor airflow simulation.

Keywords: supply opening model; numerical simulation; CFD; air jet; boundary conditions

1. INTRODUCTION

Recent advances in computational fluid dynamics (CFD) and computer power make it possible to accurately predict some features of airflow within ventilated spaces. However, there are still some significant questions for actual application, whereas CFD has become a powerful tool for indoor distribution analysis. The velocity level in a room ventilated by jet ventilation is strongly influenced by the supply conditions. The supply jets control air movement in the room and, therefore, it is very important that the inlet conditions and the numerical method can generate a satisfactory description of this momentum flow. Unfortunately, supply openings applied in heating, ventilating and air conditioning (HVAC) industry usually have
complicated geometry. Since 1970’s, some models have been developed to describe the inlet boundary conditions of supply openings for indoor airflow simulation [1]. Generally, the existing supply opening models may be divided into two types: models describing boundary conditions in the vicinity of supply openings, and the others converting the description of boundary conditions to the description of a box boundary conditions around the supply opening. The former describes the mass and momentum boundary conditions directly while the latter describes those conditions on the surfaces of the box, which is an indirect one.

Indirect supply opening model includes box method [1][2], prescribed velocity method [3] and main region specification method [4]. Box method transfers the description of the diffuser boundary conditions to the box boundary conditions. The boundary conditions at the box surface parallel to the supply opening surface are measured data, and the boundary conditions at the box surface perpendicular to the supply opening surface are considered as: $\frac{\partial \phi}{\partial n} = 0$, where $n$ is the direction parallel to the studied surface and, $\phi$ is velocity, kinetic energy and the dissipation rate of kinetic energy etc. Comparisons between simulated and measured data shows that box method can get good result [1]. But it demands detailed measurement for each supply opening, which is not practical in engineering application. Similar with box method, prescribed velocity method describes the boundary conditions at the box surfaces around the supply opening. In this method, the inlet boundary conditions are specified using conventional method; nevertheless, one component of the velocity is measured inside the assumed box area. The velocity component values inside the box are prescribed as extra boundary conditions to correct the predicted velocity around the inlet area [3]. Good results can also be gotten by the method. However, measurements for each supply opening are still needed. Another way to apply this method is adopting the jet formulas to calculate the velocity component instead of measurements. But because of the complicated conditions in the real room, the jet formulas can not predict the velocity around the box correctly. Some examples shown in reference [5] indicate the uncertainty of jet formulas. Another indirect supply opening model is the main region specification method [4][6]. This model takes advantage of diffuser characteristic equations. The specification of inlet boundary is converted to the specification of the surfaces of a volume around the diffuser. One volume surface is located inside the main region of the diffuser jet. But the examples of reference [5] show that, jet formulas of diffuser characteristic are not proper to predict indoor air distribution, therefore, main region specification method is not applicable. Furthermore, it may cause much error for those cases in which the diffuser jet is strongly confined, as the jet is not upstream influence.

Unlike the methods based on box, models describing inlet boundary conditions directly need neither measurement nor empirical formulas. In principle, direct supply opening models can be applied for any cases, no matter the diffuser jet is confined or non-isothermal. So it is strongly recommended to apply this kind of supply opening model for engineering application. There are mainly two direct supply opening models: basic model [7] and momentum model [8]. Basic model uses a simple
opening with the same effective area as the complex diffuser to simulate inlet momentum fluxes correctly. This model is a coarse method, even to those supply openings whose effective area is small. In actual application, there are many supply openings of this kind, such as perforated panel, displacement ventilation diffusers, etc. The example of reference [9] shows that basic model may cause much error for HESCO diffuser, which has small effective area [9]. Momentum model describes velocity vector based on diffuser effective area to describe the velocity correctly. To keep the appropriate supply air mass fluxes and to introduce the same amount of air into the room, the boundary conditions for continuity equation and momentum equations are separately described [8]. So it is not convenient to use this method as most commercial CFD software or usual CFD algorithm do not support the separate description of boundary conditions for continuity and momentum equations. And the most important is that momentum method does not consider the cases that air jet has an angle with the diffuser. For example, the louvers (vanes or bars) of a grille are usually set to produce a wide horizontal spread of the stream, in which the outlet jet is diverged. Momentum method does not tell how to describe the boundary conditions of this case. And basic model can not solve it, either. Therefore, a new method called N-point supply opening model is developed to solve the problems, which also describes the boundary conditions directly [10]. The new method models the inlet momentum, mass and buoyancy flow rate correctly without separate description of boundary conditions of the equations. Although N-point supply opening model improves the description of inlet boundary conditions relatively, it can not solve too complicated problems [10].

However, the most reliable method is full representation of supply openings. Since it relies on neither jet types (free or confined, isothermal or non-isothermal) nor empirical jet equations, full representation method is the most universal model to describe inlet boundary conditions for indoor airflow simulation. Chen and Moser simulated HESCO diffuser by full representation method. The results were reasonable [8]. Emvin and Davidson also calculated the penetrating length of HESCO diffuser under cooling conditions by the method, and the results agreed well with measured data [9]. Although full representation of supply opening is rather expensive, we can develop new model based on it to make use of its advantage. As grilles are most commonly used in engineering, and there are complicated inlet conditions actually, some cases are studied to validate the full representation method while some results by N-point supply opening model are also presented.

2. SIMULATION OF GRILLE AIR JETS

2.1 Cases selection

A commercial grille, QINGYUN FK-2A 250-550, is studied. It is 550 mm in width and 250 mm in height, having 20 louvers and 21 spaces, each space is 15 mm wide. The effective area of the grills is about 65% of the gross area. See figure 1.
There are 4 outlet conditions shown in the manual [11]:

![Outlet conditions](image)

Figure 2 The four outlet conditions of the grille

For case A, the louvers are parallel and the outlet momentum has one direction; the louvers of case B, C and D are set to produce a wide horizontal spread of the stream, and there are 3, 5 and 7 directions respectively. The inlet air volume is set to 480 m³/h.

The geometry of simulated room is: \( L_R \times H_R \times W_R = 10m \times 3m \times 6m \), and the perforated panels are located in the middlemost of the west wall, which ensure the supply air jet is free [12]. And there are no difference between supply and indoor air temperature to generate isothermal free air jet, since isothermal free air jet is the basic of diffuser characteristic.

### 2.2 Simulation software

STACH-3, a three-dimensional CFD software developed by Li and Zhao, was applied for this study [13]. As \( k - \varepsilon \) turbulence model is proper for isothermal indoor airflow [14][15] [16], it is applied here for free isothermal air jet. Therefore, it can be regarded that the error is caused mainly by supply opening model, not turbulence model. Then we can examine if the supply opening model is proper for describing inlet boundary conditions.

The equations based on \( k - \varepsilon \) turbulence model for STACH-3 in uniform format are as following and details are listed in table 1:

\[
\frac{\partial}{\partial t} (\rho \phi) + \text{div}(\rho \vec{u} \phi - \Gamma \phi \text{grad} \phi) = S_\phi \quad (1)
\]
<table>
<thead>
<tr>
<th>$\Phi$</th>
<th>$\Gamma_{\Phi}$</th>
<th>$S_{\Phi}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$</td>
<td>$\mu_{\text{eff}}$</td>
<td>$-\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial x}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial x}) + g_z (\rho - \rho_{\text{ref}})$</td>
</tr>
<tr>
<td>$v$</td>
<td>$\mu_{\text{eff}}$</td>
<td>$-\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial y}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial y}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial y}) + g_y (\rho - \rho_{\text{ref}})$</td>
</tr>
<tr>
<td>$w$</td>
<td>$\mu_{\text{eff}}$</td>
<td>$-\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} (\mu_{\text{eff}} \frac{\partial u}{\partial z}) + \frac{\partial}{\partial y} (\mu_{\text{eff}} \frac{\partial v}{\partial z}) + \frac{\partial}{\partial z} (\mu_{\text{eff}} \frac{\partial w}{\partial z}) + g_z (\rho - \rho_{\text{ref}})$</td>
</tr>
<tr>
<td>$k$</td>
<td>$\frac{\mu_{\text{eff}}}{\sigma_k}$</td>
<td>$G_k - \rho \varepsilon$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>$\frac{\mu_{\text{eff}}}{\sigma_\varepsilon}$</td>
<td>$\frac{\varepsilon}{k} [G_k C_1 - C_2 \rho \varepsilon]$</td>
</tr>
</tbody>
</table>

\[ \mu_{\text{eff}} = \mu_1 + \mu_i \quad \mu_i = C_D \rho k^2 / \varepsilon \]
\[ G_k = \mu, \{2[(\frac{\partial u}{\partial x})^2 + (\frac{\partial v}{\partial y})^2 + (\frac{\partial w}{\partial z})^2] + (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})^2 + (\frac{\partial w}{\partial z} + \frac{\partial v}{\partial y})^2 + (\frac{\partial u}{\partial x} + \frac{\partial v}{\partial z})^2 \} \]
\[ C_1 = 1.44, C_2 = 1.92, C_D = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3 \]

Where, $x, y, z$ is the three Cartesian coordinates and $u, v, w$ is the velocity component in $x, y, z$ direction respectively, $k$ is the turbulent kinetic energy, $\varepsilon$ is the dissipation rate of the turbulent kinetic energy, $p$ is the effective pressure, $\rho$ is the density of air, which is a constant because indoor air can be considered as incompressible, $\mu_1$ is the kinematic viscosity of air, $\mu_i$ is the eddy viscosity of turbulence, $\mu_{\text{eff}}$ is the effective viscosity of turbulent air.

Finite volume method is applied in STACH-3, and the momentum equations are solved on staggered grid. The difference scheme is hybrid scheme. Wall functions are adopted to consider the effects of walls.

### 2.3 Analysis of results

For case A and B, the simulated centerline velocity and measured data from reference [17][18] are shown in figure 3. They agree well with each other. The centerline velocity of this kind of grille can be calculated by the following formula [17][18]:

$$ \frac{V_m}{V_0 \sqrt{C_d R_{fa}}} = K \frac{\sqrt{A_c}}{X} $$  \hspace{1cm} (2a)  

$$ \frac{V_m}{V_0} = K \frac{\sqrt{A_c C_d R_{fa}}}{X} = K \frac{\sqrt{A_0}}{X} $$  \hspace{1cm} (2b)  

$V_m$: centerline velocity, m/s;  
$V_0$: supply air velocity, m/s;  
$C_d$: coefficient of discharge, 0.65~0.9;  
$R_{fa}$: coefficient of effective area;  
$K$: proportion constant, decided by measurement;  
$A_c$: gross area of the supply opening, m$^2$;
\( X \): distance from the supply opening face, m; \( A_0 \): effective discharge area, m²; 
\( A_0 = A_1 C_4 R_b \).

The simulated and measured \( K \) are shown in table 2. The comparison shows that the simulated results are reasonable. And the simulated results also give the self-preserving distribution of cross-section velocity, which indicates the full establishment of turbulence in main region, agreeing well with the experiment [17]. The other cases of various inflow air volume have similar results and they are omitted here.

### Table 2 Comparison of centerline velocity coefficient \( K \) (Case A and B)

<table>
<thead>
<tr>
<th>Case</th>
<th>Simulated ( K )</th>
<th>Measured ( K )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>4.72</td>
<td>4.7(^{[17]}), 4.8(^{[18]})</td>
</tr>
<tr>
<td>B</td>
<td>3.23</td>
<td>3.2–3.5(^{[17]})</td>
</tr>
</tbody>
</table>

![Graphs](image)

a) case A  

Figures 3  

Comparison of centerline velocity

b) case B

For case C and D, the manufacturer’s catalogs give the jet range as the important character of the grille. Here the jet range is the jet spread length where the centerline velocity reaches 0.5 m/s. To test the universality of simulated results, four different supply air volumes are considered. To compare the results by full representation method with other supply opening model, the jet ranges of case C and D are also simulated by N-point supply opening model. The results of the two models and measured data are presented in table 3.

### Table 3 Comparison of jet range (Case C and D)

<table>
<thead>
<tr>
<th>Case</th>
<th>Supply air volume (m³/h)</th>
<th>Jet range by N-point model (m)</th>
<th>Jet range by full representation (m)</th>
<th>Jet range by experiment(^{[11]}) (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>480</td>
<td>1.15</td>
<td>2.30</td>
<td>2.45</td>
</tr>
<tr>
<td></td>
<td>960</td>
<td>2.00</td>
<td>4.12</td>
<td>4.40</td>
</tr>
<tr>
<td></td>
<td>1440</td>
<td>3.70</td>
<td>5.21</td>
<td>5.89</td>
</tr>
<tr>
<td></td>
<td>1920</td>
<td>5.10</td>
<td>6.23</td>
<td>6.72</td>
</tr>
<tr>
<td>D</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>480</td>
<td>0.75</td>
<td>1.53</td>
<td>1.54</td>
</tr>
<tr>
<td></td>
<td>960</td>
<td>1.40</td>
<td>2.50</td>
<td>2.66</td>
</tr>
<tr>
<td></td>
<td>1440</td>
<td>1.71</td>
<td>3.22</td>
<td>3.61</td>
</tr>
<tr>
<td></td>
<td>1920</td>
<td>1.85</td>
<td>4.12</td>
<td>4.32</td>
</tr>
</tbody>
</table>

It is obvious that the simulated jet ranges are very close to measured data, while
the results by N-point supply opening model are shorter. The poor results may be caused by the simplicity of actual complicated outlet conditions. However, full representation method describes the 21 spaces of the grille, which is more close to the real outlet conditions.

3. CONCLUSION

Examples of grille under different outlet conditions prove that the full representation of supply opening is trusty to describe the inlet boundary conditions for indoor airflow simulation. For relative simple outlet conditions (case A and B), the modeled jet characteristics agree well with experiments. For rather complicated outlet conditions (case C and D), the simulated jet ranges by full representation method are close to the measured data. As full representation costs too much time, some new model with higher efficiency may be developed based on it since the method is so accurate and universal.

References


