

Chen, Q., Lee, K., Mazumdar, S., Poussou, S., Wang, L., Wang, M., and Zhang, Z. 2010. "Ventilation performance prediction for buildings: Model Assessment," *Building and Environment*, 45(2), 295-303.

Ventilation performance prediction for buildings: Model Assessment

Qingyan Chen^{a,b,*}, Kisup Lee^b, Sagnik Mazumdar^b, Stephane Poussou^b, Liangzhu Wang^b, Miao Wang^b, and Zhao Zhang^b

^a School of Environment Science and Technology, Tianjin University, 92 Weijin Road, Nankai District, Tianjin 30072, China

^b National Air Transportation Center of Excellence for Research in the Intermodal Transport Environment (RITE), School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA

Phone: +1-765-496-7562, Fax +1-765-494-0539, E-mail address: yanchen@purdue.edu

ABSTRACT

Designing ventilation systems for buildings requires a suitable tool to assess the system performance. This investigation assessed seven types of models (analytical, empirical, small-scale experimental, full-scale experimental, multizone network, zonal, and CFD) for predicting ventilation performance in buildings, which can be different in details according to the model type. The analytical model can give an overall assessment of a ventilation system if the flow could be approximated to obtain an analytical solution. The empirical model is similar to the analytical model in terms of its capacities but is developed with a database. The small-scale model can be useful to examine complex ventilation problems if flow similarity can be maintained between the scaled model and reality. The full-scale model is the most reliable in predicting ventilation performance, but is expensive and time consuming. The multizone model is a useful tool for ventilation design in a whole building, but cannot provide detailed flow information in a room. The zonal model can be useful when a user has prior knowledge of the flow in a room. The CFD model provides the most detailed information about ventilation performance and is the most sophisticated. However, the model needs to be validated by corresponding experimental data and the user should have solid knowledge of fluid mechanics and numerical technique. Thus, the choice for an appropriate model is problem-dependent.

1. Introduction

Designing ventilation systems for buildings requires a suitable model to assess system performance. The performance can be assessed in terms of airflow pattern, the distributions of contaminant concentration, air velocity, and air temperature, global ventilation effectiveness and air distribution effectiveness, etc. We have recently conducted a review of the available models for predicting ventilation performance in buildings [1]. The review was focused on the contributions of the models to the literature. Although the review provided basic background for each type of models, it did not assess the pros and cons of the models for assessing ventilation performance in buildings. This paper intends to assess these models further by applying them to various ventilation problems in buildings. The results may provide designers a guide for

* Authors' names appear alphabetically.

selecting a suitable model for designing comfortable, healthy, safe, and energy-efficient ventilation systems in buildings.

2. Assessment of various types of models

Our review [1] of the models for predicting ventilation performance in buildings identified seven types of models - analytical, empirical, small-scale experimental, full-scale experimental, multizone network, zonal, and CFD. To assess the performance of the models, it may not be wise to apply them to only one ventilation system, since each model type has its own limitations on the applications. Thus, our study would assess each model type by selecting a suitable ventilation system that would be most favorable to them.

2.1 Analytical models

The analytical models are not used to seek the most accurate and detailed flow information of a ventilation system. Instead, the models are used to gain a quick estimate of the ventilation performance. These models use major assumptions so that an analytical solution can be obtained and the solution is expected to reflect the most important feature of the system performance.

Let us consider the performance of a ventilation system in a workshop for its ability to remove an assumed contaminant released at a constant rate of 10^{-6} kg/s in a position shown in Fig. 1. The workshop was 40 m long with 50 workers. Clean air was supplied using two linear overhead inlets in the side walls and was extracted by two outlets on the sidewalls at the floor level. The supply airflow rate was 10 L/s per worker. The linear ventilation system would create strong mixing in the cross sections, so it would be acceptable to assume the contaminant distribution in the cross section to be uniform. Thus, if the contaminant dispersion can be calculated along the longitudinal direction, the performance of the ventilation system could be estimated.

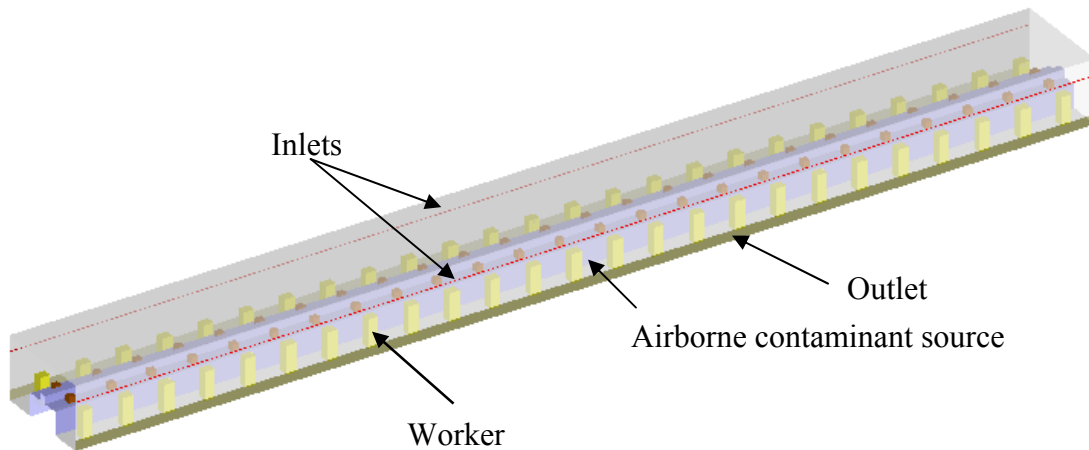


Fig. 1. Schematic of the workshop

This investigation used the analytical model developed by Mazumdar and Chen [2] for airliner cabins. This model assumed a uniform contaminant distribution in the cabin cross-section due to high air exchange rates. This assumption simplified the contaminant transport

phenomenon inside the complex cabin geometry to a one-dimensional transmission model. Mazumdar and Chen analytically solved the governing equation for contaminant transport using the principle of superposition and the method of separation of variables [3]. Considering only diffusive transport and a contaminant source of strength S_ϕ (kg/s) located at $x = L_1$ releasing from time $t = 0$ to $t = t_0$, the contaminant concentration (C) for locations $x \leq L_1$ at $t \leq t_0$ is:

$$C = C_{inlet} + A_{L_1} e^{m_1(L_1-x)} + B_{L_1} e^{m_2(L_1-x)} + a_0 e^{-\beta_0^2 t} + 2 \sum_{n=1}^{\infty} a_n \cos[\alpha_n (L_1 - x)] e^{-\beta_n^2 t} \quad (1a)$$

and for $x \geq L_1$:

$$C = C_{inlet} + A_{L_2} e^{m_1(x-L_1)} + B_{L_2} e^{m_2(x-L_1)} + a_0 e^{-\beta_0^2 t} + 2 \sum_{n=1}^{\infty} a_n \cos[\alpha_n (x - L_1)] e^{-\beta_n^2 t} \quad (1b)$$

where C_{inlet} is the contaminant concentration in the supply air. Furthermore, the contaminant concentration for $t \geq t_0$ is:

$$C = C_{inlet} + A e^{m_1 x} + B e^{m_2 x} + b_0 e^{-\beta_0^2 (t-t_0)} + 2 \sum_{m=1}^{\infty} b_m \cos(\alpha_m x) e^{-\beta_m^2 (t-t_0)} \quad (2)$$

The coefficients ($A, B, a, b, \alpha, \beta$) depend on the geometry of the cabin and are calculated using the boundary conditions at the source location and at the end walls [2].

This workshop could be simplified in the same way as an airliner cabin is. Thus, the analytical model could be used here to obtain the contaminant concentration under steady state along the longitudinal direction of the workshop, as shown in Fig. 2(a). The corresponding results obtained by CFD were used for comparison. The contaminant concentration profiles agree with each other. The CFD computations took about one day of computing time using a 10-node computer cluster while the analytical model gave instant results.

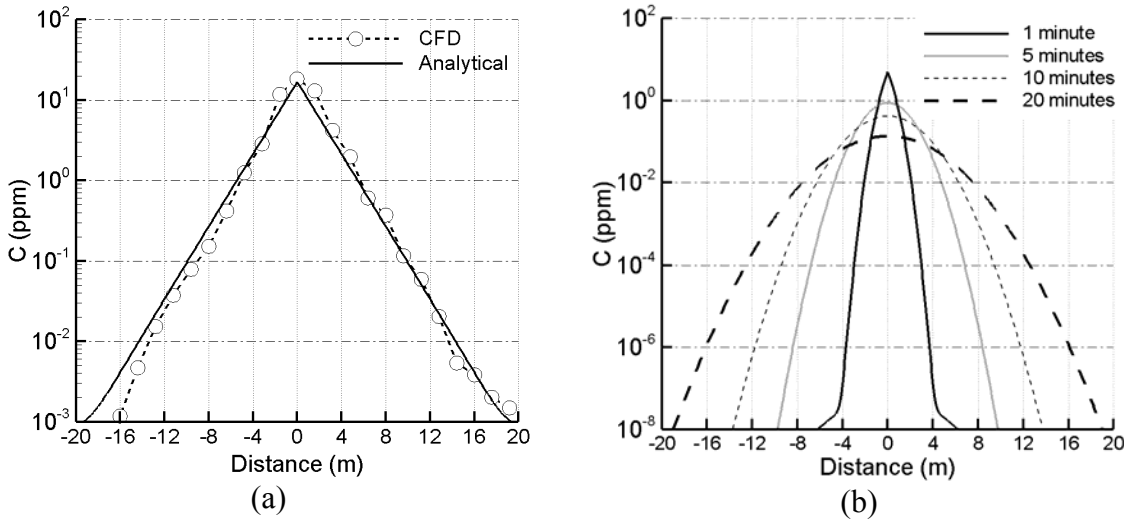


Fig. 2. Contaminant transport along the longitudinal direction in the workshop: (a) Comparison between the analytical and CFD models and (b) Contaminant dispersion for a one-minute contaminant release

The analytical model was also used to study transient contaminant dispersion by assuming that the contaminant was released at the source location for only one minute. Fig. 2(b) shows the transient profiles of the contaminant concentration at $t = 1, 5, 10$ and 20 min, respectively, after the contaminant was released in the workshop. The results illustrate that the ventilation system can slowly reduce the contaminant concentration level. Roughly, the peak contaminant concentration was reduced by two orders of magnitude in 20 minutes.

The analytical model can provide global information of the contaminant transport with little computational effort, though it may not provide the details in a cross-sectional plane. If the workshop could not be simplified to a one-dimensional problem, the analytical model cannot be used. If one has to develop an analytical model, it is not a trivial job. The model developed by Mazumdar and Chen requires considerable knowledge of mathematics and engineering skills to simplify the problem.

2.2 Empirical models

The empirical models could be used to predict the performance of a ventilation system in a very similar way as the analytical models do. The analytical and empirical models do not differ very much in their principles. However, the empirical models are often developed not only from the conservation equations of mass, energy, and chemical species, but also from data of experimental measurements or advanced computer simulations.

In order to assess the capacity of an empirical model for predicting the performance of a ventilation system, this study used air distribution effectiveness as a criterion for performance. The air distribution effectiveness, E , is defined as

$$E = \frac{C_e - C_s}{C - C_s} \quad (3)$$

where C_e is the contaminant concentration at ventilation exhaust, C_s is the contaminant concentration at the supply, and C is the contaminant concentration in the breathing zone of the room. Lee et al. [4] used a database to develop the following equation for calculating the air distribution effectiveness, E , through a statistical analysis method. The database consisted of the air distribution effectiveness for hundreds of cases of traditional displacement ventilation and underfloor air distribution systems for six types of indoor spaces.

$$E = 1.9 + 0.000257CFG + 0.0105 BD + 0.000008BCD - 0.358B - 0.00025BC + 0.00591AF \quad (4)$$

where A is the diffuser type (1 for underfloor air distribution, 2 for traditional displacement ventilation), B is the total flow rate (ACH), C is the flow rate per diffuser (m^3/h), D is the supply air temperature ($^{\circ}\text{C}$), F is the cooling load (W/m^2), and G is the diffuser density ($\#/\text{m}^2$).

This study used Equation (4) to estimate the air distribution effectiveness at the breathing zone for 27 cases of different types of spaces (offices, workshops, and auditoriums) under various thermo-fluid boundary conditions. The thermo-fluid boundary conditions can be expressed in terms of A , B , C , D , F , and G in Equation (4). Fig. 3 shows that the E at the breathing zone can be very well calculated by Equation (4), compared with the E in the database of Lee et al. [4]. In most cases, the difference between the database and the predicted E is less than 10%. The difference is certainly acceptable for predicting ventilation system performance. Of course, Equation (4) is valid within the database. This is one of the limitations of the empirical models.

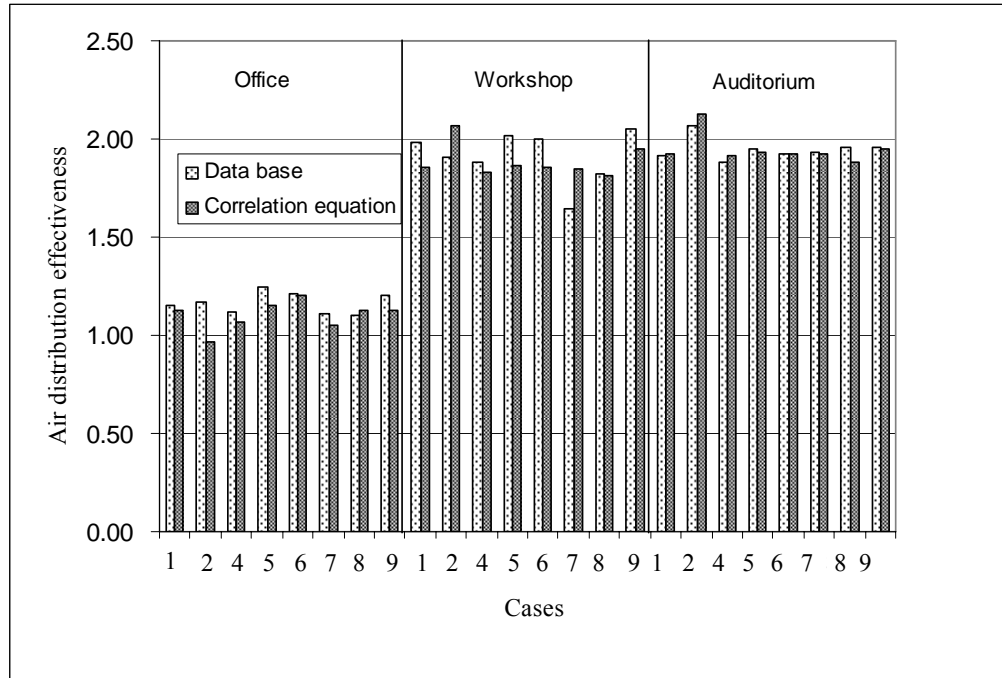


Fig. 3. Comparison of the air distribution effectiveness obtained with empirical equation (4) and from the database [4]

Again, similarly to the analytical models, the empirical models could only produce very limited information about ventilation performance in buildings. Normally, empirical models are developed with a lot of approximations so that they are not universal. However, the models are generally very simple and easy to use. The results obtained by empirical models are often sufficient for ventilation system design.

2.3 Small-scale experimental models

As indicated in our review [1], the performance of ventilation systems may be evaluated through adequately scaled experimental models, which can provide more practical temporal and spatial resolutions than full-scale models. Scaled models can also incorporate complex flow, geometric and thermal conditions, unlike analytical and empirical models. In order to represent correctly the ventilation flow in an actual room or building, small-scale models should be designed following similarity analysis, in which the relevant dimensionless flow parameters are identical between small- and full-scale (*e.g.* Reynolds, Grashof, Prandtl numbers).

To illustrate the capacities of a scaled model, our investigation [5] used a one-tenth scale water model to investigate the effects of a moving body on airflow and contaminant transport inside an airliner cabin, whose seat configuration resembles a packed auditorium, conference room, or classroom. Fig. 4 shows the scaled experimental test facility with an upside-down cabin mockup fully submerged in an index-matching water tank. The cabin wall was semi-circular with an inside diameter of 0.45 m and a length of 2.43 m, which was equivalent to 28 rows of economy-class seats in commercial aircraft. This study did not consider the secondary effects of seats or other obstructions, in order to focus on fundamental interactions between wake and ventilation pattern. A moving passenger or crew member was simulated by a block of 0.02 m thick \times 0.05 m wide \times 0.17 m tall, which translated longitudinally along the cabin centerline. In

practice, it is usually impossible to match the Reynolds and Grashof numbers simultaneously; therefore isothermal conditions were used in this study, allowing the airliner cabin to be upside-down due to the absence of buoyancy effects.

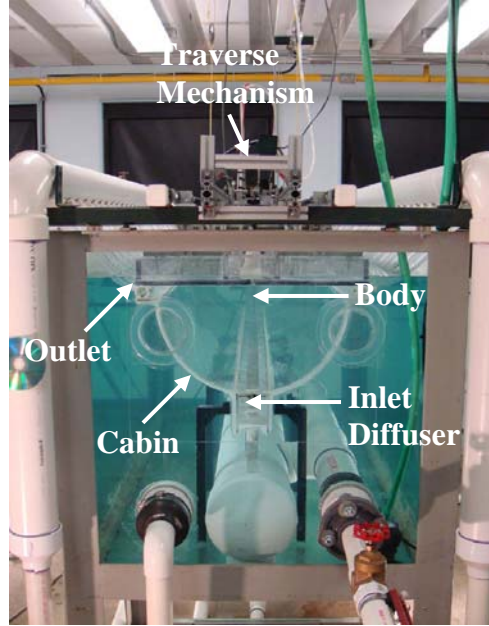


Fig. 4. The small-scale experimental test facility of an airliner cabin in a water tank

Using the Particle Image Velocimetry (PIV) technique, the cross-sectional flow could be quantitatively measured. Fig. 5 shows the flow downstream of the moving body at a distance equivalent to 1.5 body width from a fixed cross-sectional plane. Vertical symmetry in the wake, discernable in the instantaneous velocity field of Fig. 5(a), is evidenced by the mean velocity field of Fig. 5(b), obtained by phase-averaging one hundred measurements.

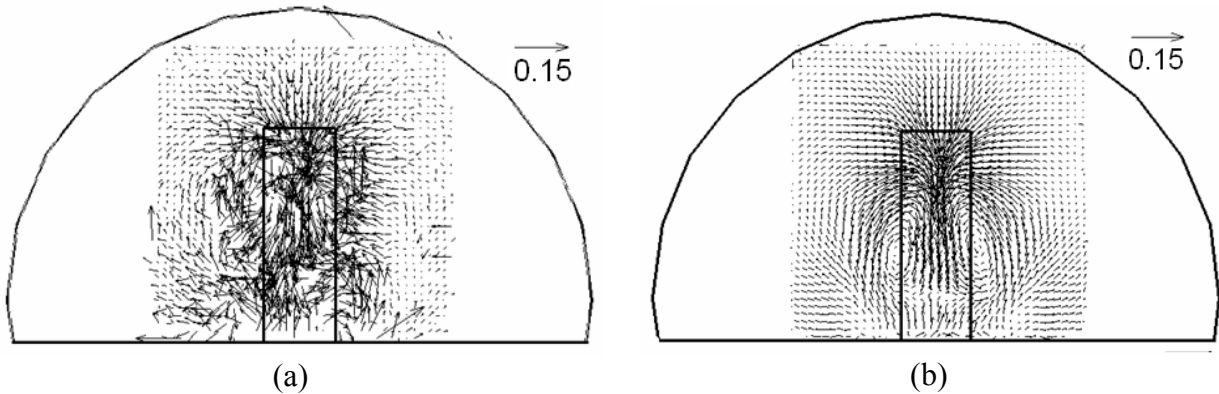


Fig. 5. (a) Instantaneous and (b) phase-averaged velocity fields obtained using the PIV technique

Contaminant transport inside the small-scale airliner cabin could also be visualized using the Planar Laser-Induced Fluorescence (PLIF) technique. Such qualitative measurements use a fluorescent dye injected into the flow and illuminated by a laser sheet [6]. In our study, the dye was released from the lateral sides of the moving body. Fig. 6 shows a typical instantaneous image taken using the PLIF technique after the body moved through a cross-sectional plane

(downstream distance equivalent to eight body width). Contaminant mixing with the ambient fluid is visible through the variations in light intensity.

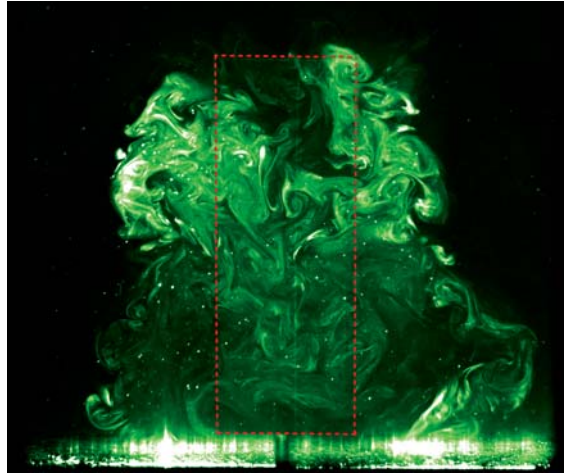


Fig. 6. Instantaneous cross-sectional flow visualization of contaminant migration using the PLIF technique

Such quantitative and qualitative results complement analytical and empirical models, by providing detailed characteristics of ventilation performance under a variety of conditions at lower economical cost than full-scale models. However, small-scale experimental modeling has its limitations. First, scaling of the flow by similarity may not permit an exact match of the dimensionless flow parameters. Second, it may be difficult to construct geometrically-complex air-supply diffusers commonly employed in actual rooms or buildings. Last, accurate characterization of small-scale flows requires a relatively significant investment in time, equipment and experimental expertise.

2.4 Full-scale experimental models

Because a small-scale experimental model often has difficulty achieving flow similarity, a full-scale experimental model is widely used. A full-scale model typically models a room that could be simulated by an environmental chamber or a house/apartment. By using an environmental chamber, the thermo-fluid boundary conditions can be controlled. However, it is not easy to control the boundary conditions in a house or an apartment. A full-scale model can simulate the most realistic conditions. However, the construction of a full-scale experimental model is not always feasible. For example, if one wishes to evaluate ventilation performance in a gymnasium, it is not feasible to build a full-scale mockup because of its size and costs.

This investigation used an environmental chamber of $2.44\text{ m} \times 2.44\text{ m} \times 2.44\text{ m}$ to simulate an occupied space, as shown in Fig. 7. The chamber was used to obtain high quality experimental data of the distributions of air velocity, temperature, and turbulence for validating CFD models. The study was for indoor airflow in a typical room with furniture, heat sources, and an overhead air-supply diffuser. To eliminate possible errors that may be introduced from complex geometry, this investigation simplified the room features. For example, a large box of $1.22\text{ m} \times 1.22\text{ m} \times 1.22\text{ m}$ was placed on the floor in the center of the room to simulate furniture in the room. A heat source of 700W was placed in the box to simulate heat generated from the

occupants and equipment in the room. A complex air-supply diffuser was replaced by a simple slot opening on the upper left corner along the whole width of the room.

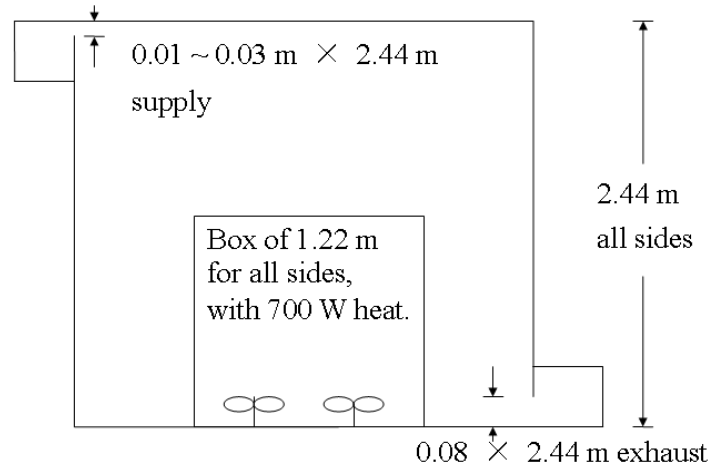


Fig. 7 The sketch of the test room with a box, heat source, and air supply and exhaust

The measurements of the air velocity, temperature, and turbulence could be obtained by using PIV or anemometers. The use of PIV for such a space would not be easy, due to the space needed to project the laser light, the large size of the room, and the difficulty in maintaining the thermal boundary conditions if transparent material were used as walls. Thus, this study used ultrasonic anemometers to measure the air velocity, temperature, and turbulence distributions. Our ultrasonic anemometers have a small sensor, so they are expensive. Each of them costs around \$15,000 USD.

Fig. 8 shows only the air velocity and temperature distributions in the mid-streamwise section. Due to the presence of the box, the airflow was fully three dimensional with many recirculations. The heat did general a thermal plume but was not very strong.

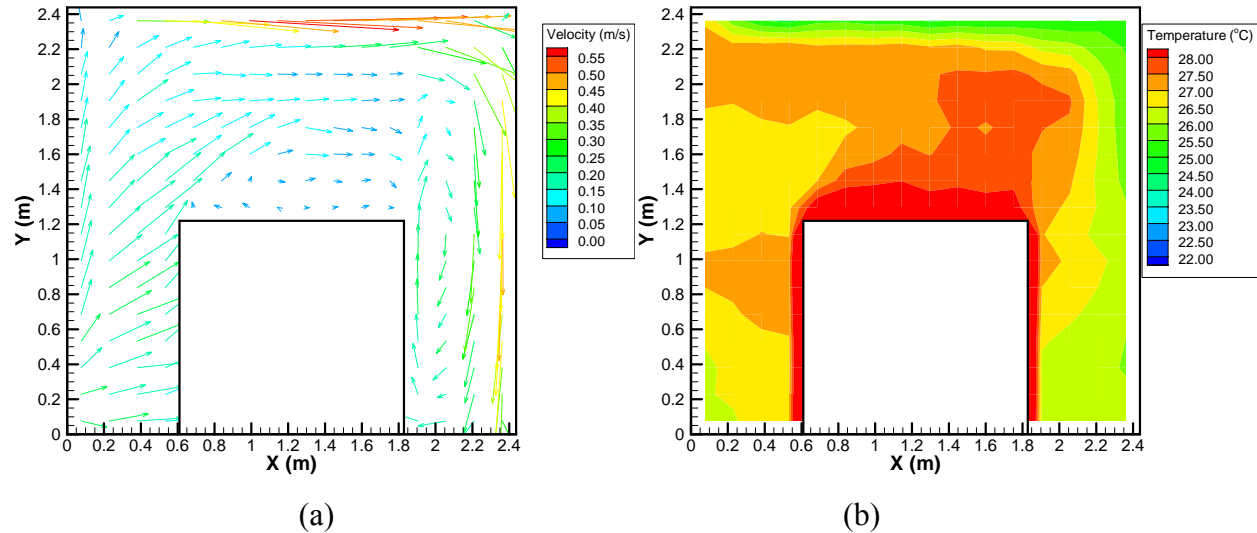


Fig. 8 The air velocity and temperature distribution in the mid-streamwise section of room: (a) air velocity and (b) air temperature

Clearly, measurements obtained from a full-scale experimental model are the most realistic and the data obtained is most reliable. Such detailed flow information cannot be obtained with analytical and empirical models. A full-scale experimental model does not have the scaling problem associated with the small-scale experimental model. In addition, the results obtained are the most straightforward. Unfortunately, a full-scale experimental model with necessary measuring and control equipment for assessing ventilation performance can easily cost a couple hundreds of thousands of dollars. The experiment can take a long time to complete. For instance, we had only two ultrasonic anemometers available. In order to have a resolution of 0.18 m \times 0.18 m in only two sections, a resolution essential for validating a CFD model, the measurements required a few days to complete. In addition, the experiment is not free from errors. More detailed discussion of the pros and cons of full-scale experimental models can be found in references [7-8].

2.5 Multizone airflow network models

For predicting ventilation performance in a moderately- or large-sized building, only numerical modeling by a multizone airflow network model could give the required information at a reasonable cost. The experimental measurements with small-scale and large-scale mockups are not realistic in terms of their costs. The analytical and empirical models would not produce useful information for ventilation performance analyses. A CFD model, to be presented in section 2.7, would need days of computing time for such a building, which makes the CFD model not attractive as a design tool.

A multizone model, such as CONTAM [9], calculates airflow and contaminant distributions between the zones (or rooms) of a building and between the building and the outdoors. If airflow path ij connects zone i and zone j and F_{ij} is the airflow rate through path ij , the F_{ij} can be calculated with a power-law function of the pressure drop, ΔP_{ij} , across path ij :

$$F_{ij} = \alpha_{ij} c_{ij} |\Delta P_{ij}|^{n_{ij}} \quad (5)$$

where, c_{ij} is the flow coefficient ($\text{kg/s} \cdot \text{Pa}^{n_{ij}}$), n_{ij} is the flow exponent of path ij , $\alpha_{ij} = (P_i - P_j)/|P_i - P_j|$, and P_i and P_j is the total pressure (Pa) at either zone i or j side of path ij , respectively. One of those common power-law functions used in CONTAM is the orifice equation:

$$F_{ij} = \alpha_{ij} C_D A_{ij} \sqrt{2\rho |\Delta P_{ij}|} \quad (6)$$

where, C_D is discharge coefficient, dimensionless; A_{ij} is the cross-sectional area of an opening (m^2), and ρ is air density (kg/m^3).

This study used a laboratory in a university to study the airflow and contaminant transport under steady state conditions in a moderately-sized building. Fig. 9 shows the floor plane of the laboratory, which had 64 rooms/zones and was mechanically ventilated by six air handling systems. The airflow through the flow paths is also shown in Fig. 9(a). This simulation assumed that most of the airflow paths at the building façade were closed except Window 1 and Door 1. Thus, the flow through the cracks on the façade was minimal. The internal zones were mechanically-ventilated and the airflows were therefore balanced. Due to the air mass conservation, a large quantity of flow entered through the upwind door (Door 1) and left through the downwind window (Window 1). The return vent took care of excessive mechanically-supplied air in the laboratory.

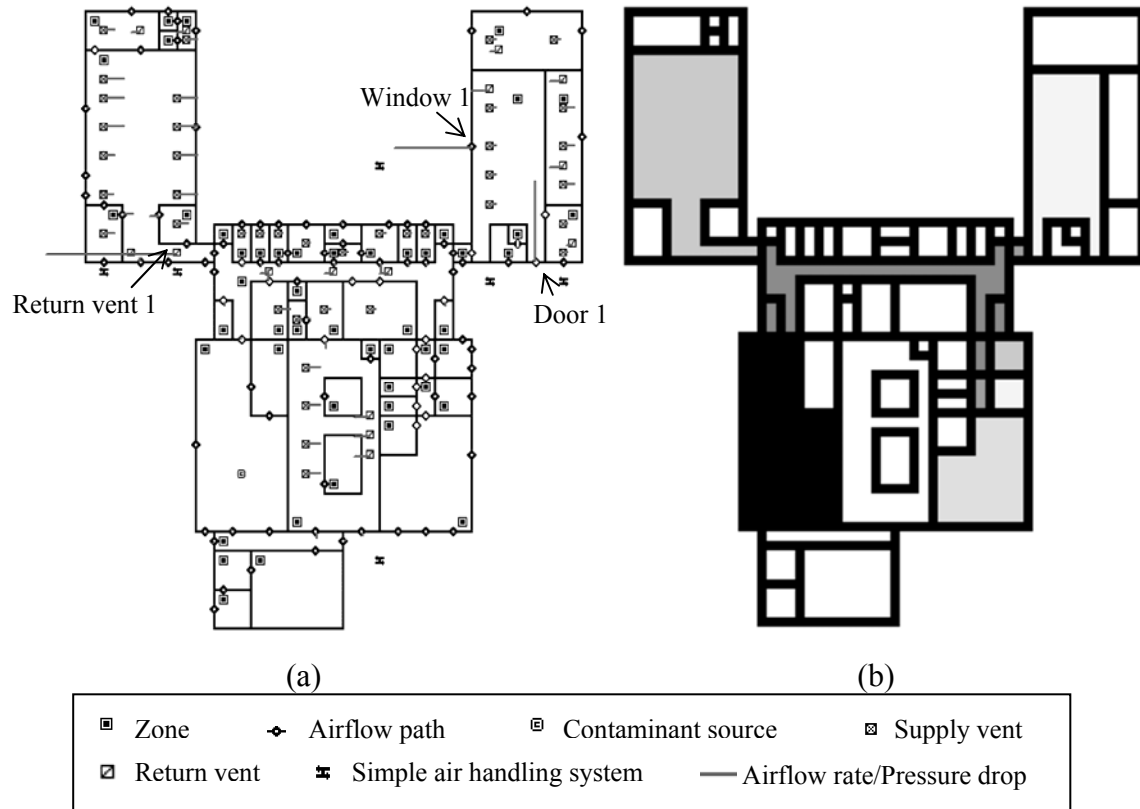


Fig. 9. The floor plan of the laboratory and (a) airflow and (b) contaminant concentration as calculated by CONTAM

Fig. 9(b) shows the contaminant concentration distribution with the source generation in the lower left hall, where the darker the color, the higher the level of contaminant concentration. The multizone model determined the contaminant dispersion between zones by the airflow. The contaminant concentration calculated could be zero for zones with high pressure.

A multizone network model seems to be the only acceptable model for analyzing ventilation performance for moderately- and large-sized buildings. The multizone models can quickly obtain ventilation performance results. However, due to the assumptions used, this model is still limited to the predictions of the average characteristics of airflows and contaminant distributions. The assumptions include the uniform air temperature and contaminant distributions in a zone, neglect of momentum effects, amongst others. Improvements on these models are underway [10]. These models have been validated at several levels: intermodel comparison, with data collected within the lab, and with field data [11,12].

2.6 Zonal models

Lack of detailed flow and contaminant distribution information in a zone (room) in the analytical, empirical, and multizone models has stimulated researchers' interests in remedying the problem. Another approach is using a zonal model to determine the detailed air distribution in a zone.

Haghighat et al. [13] presented a variation of the pressure-zonal model, POMA, which stands for Pressurized zOnal Model with Air diffuser. The model used a power law formulation

to determine airflow in a control volume of a zone as a function of pressure difference such as Eq. (5). The equation may vary with the orientation of the control volume face, the presence of a neutral plane, and any association with special cells. The variation can make the equation very complex.

To illustrate the capacities of a zonal model for determining detailed flow information in a zone, POMA was used to model natural convection in a sealed room. The room also had measured data obtained by Inard et al. [14]. The room size was 3.1 m × 3.1 m × 2.5 m. Fig. 10 shows the air temperature distributions in the mid-width section of the room with the left wall at 16.9°C, right wall at 33.0°C, ceiling at 28.5°C, and floor at 25.9°C. The temperatures of the front and back walls were 26.9°C and 27.3°C, respectively. Since the room was sealed with no air supply, the airflow was driven by natural convection. Although the discrepancies between the calculated and measured temperature were around 2 K in the upper right corner, the trends of the temperature profiles were well captured by POMA.

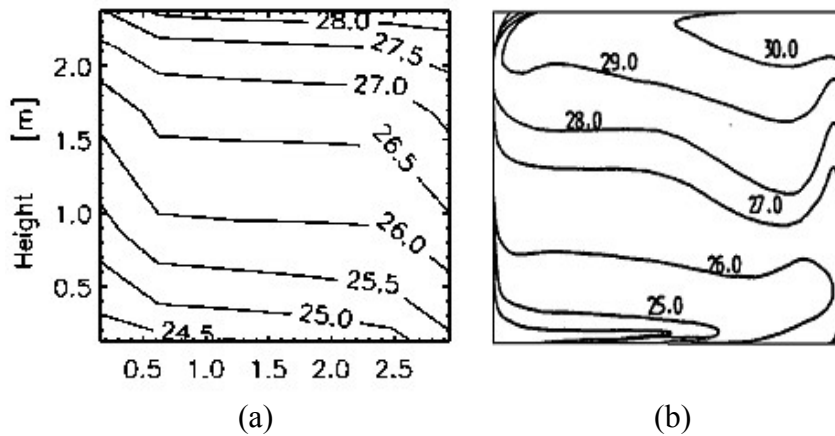


Fig. 10. Air temperature results (a) calculated by POMA and (d) measured by Inard (1996)

Indeed, a zonal model could give additional information for airflow in a zone. The information is very useful for predicting ventilation performance, especially for large zones where the air distribution cannot be assumed to be uniform. However, the case shown in Fig. 10 was the only case with a converged solution by POMA. To be successful with this model, a user should know the airflow pattern in advance in order to apply suitable equations for the special cells in the zone. If the airflow pattern could be known, one may not need the zonal model. Furthermore, the zonal model should quickly generate the results. In our case, the computing time was actually longer than that of CFD. Perhaps our experience was exceptional. Furthermore, zonal model cannot estimate the air velocity in the zone. It can only estimate the flow rate so it could not be used for evaluating thermal comfort. Nevertheless, many efforts are done to improve zonal models [15].

2.7 CFD models

With the rapid advance in computer capacity and speed, CFD models have become a powerful modeling tool for predicting ventilation performance in buildings. By solving the conservation equations of mass, momentum, energy, and species concentrations, CFD models can quantitatively present very detailed information concerning ventilation performance. Thus, the models are becoming more and more popular, as indicated in our review [1].

The most popular CFD model for predicting ventilation performance is probably the one with the RNG k- ϵ turbulence model. The turbulence model is stable and can reasonably give accurate results for most indoor airflows [16]. The CFD model solves the following transport equations:

$$\rho \frac{\partial \bar{\phi}}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{\phi}}{\partial x_j} - \frac{\partial}{\partial x_j} \left[\Gamma_{\phi, \text{eff}} \frac{\partial \bar{\phi}}{\partial x_j} \right] = S_{\phi} \quad (7)$$

where ϕ represents primary variables for ventilation performance, such as mass continuity, velocity, temperature, contaminant concentration, and turbulence parameters, $\Gamma_{\phi, \text{eff}}$ is the effective diffusion coefficient, and S_{ϕ} is the source term of an equation. The turbulence parameters solved are turbulent kinetic energy and its dissipation rate [17].

Now the question is whether CFD could be used to replace all other models mentioned in the previous sections for predicting ventilation performance. This study used a room with displacement ventilation [18] to exhibit the CFD model's capacities. Fig. 11(a) shows the room geometry with a supply diffuser, an exhaust, two occupant simulators, two computers, two tables, two pieces of furniture, and six overhead lamps. Contaminant sources were released at the head level of the two occupant simulators. Yuan et al. [18] performed detailed measurements of ventilation performance in a full-scale experimental model.

Fig. 11(b) shows the velocity distributions in two sections of the room. Since the CFD calculated the flow in the entire domain, one can see the flow information in any section. If one requires a finer resolution of the velocity distribution, it can be easily done using more numerical grids in the CFD simulation. The results indicate that the velocity magnitude was very low with room displacement ventilation. To measure low air velocity is very challenging with temperature based anemometers, such as hotwire or hot-sphere anemometers.

Fig. 11(c) and (d) depict the distributions of the air temperature and contaminant concentration in the two sections. The results are presented in continuous flood color. The air temperature and contaminant concentration were calculated with the same grid resolution as that for the air velocity. The figures used interpolation in the graphic processing. Again, the results are very informative.

However, due to the approximations used in the CFD model to make the transport equations solvable, the calculated results have uncertainties. Fig. 12 compares the computed and measured air velocity, temperature, and contaminant concentration in the center of the office. The agreement is pretty good for the air velocity and temperature, but the discrepancy is significant for the contaminant concentration. The measured air velocities by Yuan et al. [18] used hot-sphere anemometers that could not accurately measure low air velocity. It is hard to tell whether the discrepancy in air velocity near the floor was caused by the CFD model or by the measurement. The contaminant concentration data is supposed to be accurate. However, we have never obtained a good agreement between computed and measured contaminant concentration profiles in a room, even if the computations were performed by different users and the measurements were obtained for different ventilation scenarios.

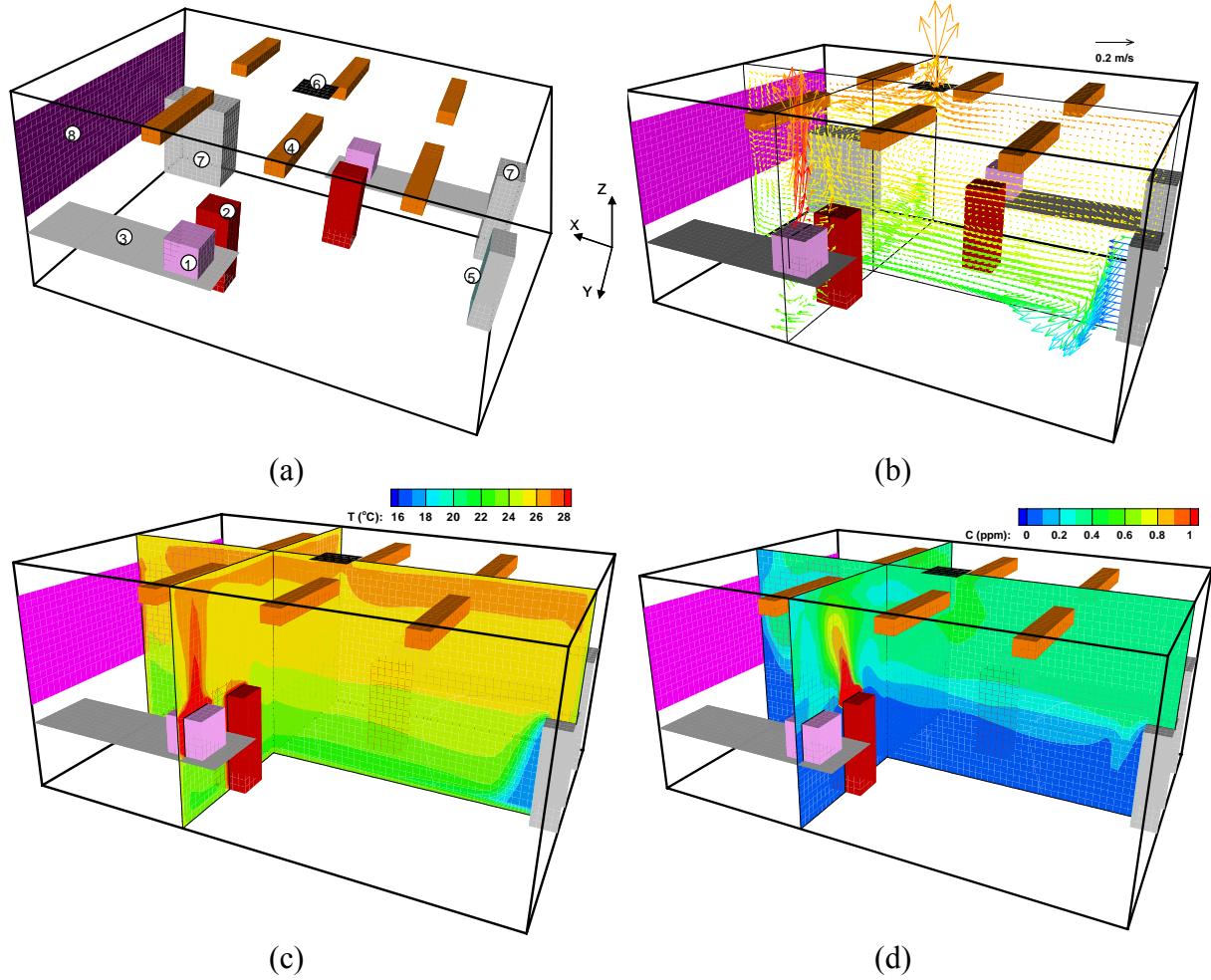


Fig. 11. (a) Sketch of an office with displacement ventilation. (1-computer, 2-occupant simulators, 3-tables, 4-lamps, 5-air supply diffuser, 6-air exhaust, 7-furniture, and 8-window); (b) Air velocity distributions; (c) Air temperature distributions; and (d) Contaminant concentration distributions

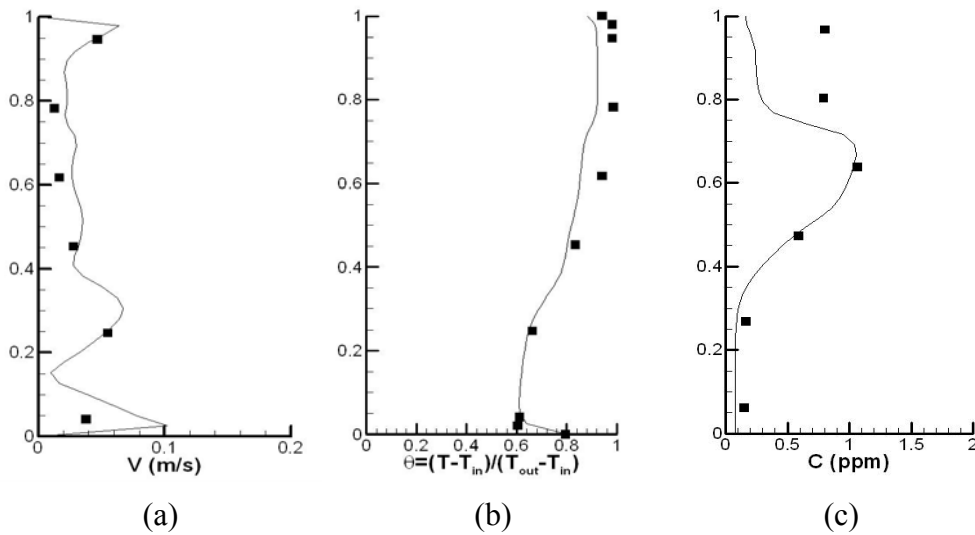


Fig. 12. Comparison of the computed (lines) and measured (square symbols) results in the center of the office along the room height: (a) air velocity, (b) air temperature, and (c) contaminant concentration

Obviously, the CFD results are the most informative, compared with the other models discussed in this paper. The CFD models are also the most accurate numerical models. However, the CFD models are probably the most difficult to use. Typically, it takes three to four months to train a graduate research assistant to use a CFD model with confidence. The research assistant also should have a good knowledge of fluid dynamics and numerical technique. Unless the research assistant could validate his/her numerical simulations by corresponding experimental data, the results may not be trusted. Our experience also shows that it is hard to get very good validation without fine-tuning the CFD modeling. For assessing the performance of a new ventilation system, a well-trained CFD user may not be able to obtain accurate results though he/she may have had experience for a different ventilation system. A new validation is always needed when the flow features change. Thus, the CFD models cannot be used to replace completely experimental measurements. Nevertheless, the CFD models can be a powerful tool for parametric studies and for optimizing ventilation system design, and are a great extension of the experimental measurements while reducing costs.

Compared with the multizone and zonal models, the CFD models typically use more computing time. A multizone model can calculate airflow in a moderately-sized building with a few seconds of computing time on a PC. A CFD model for a building with a few zones such the laboratory shown in Fig. 9 would take a few days of computing time, which is too long for a designer. Thus, each model can be used for a specific propose. For example, multizone models can be used to study the overall performance of a hybrid ventilation system. Should a CFD model be used, the computing time would be too long to be acceptable in practical use. The CFD models would also require very detailed thermo-fluid boundary conditions that may not be available in the early design stages of a ventilation system. Thus, the CFD models cannot be used to replace the multizone models; they should be combined to enhance the accuracy in the multizone models and to reduce the computing time required by the CFD models [10].

3. Conclusions

This paper reported our assessment of the seven types of models for predicting the performance of ventilation systems in buildings. The models types were analytical, empirical, small-scale experimental, full-scale experimental, multizone, zonal, and CFD.

The analytical and empirical models were normally developed with sound physics, but with a lot of approximations. Thus, it is not a trivial task to develop such models. The performance of the two types of the models was similar. The models could give quick assessment of ventilation performance with little computing effort, but detailed ventilation information was not available. The information obtained by the analytical and empirical models is often sufficient for ventilation system design.

The small-scale and full-scale experimental models could give more detailed information about the performance of ventilation systems than the analytical and empirical models. The results obtained with the experimental measurements were the most reliable, so they were used to validate the numerical models. The experimental measurements are generally expensive, especially with a full-scale model, compared with other models studied in this investigation. There are often scaling problems while trying to achieve flow similarity between a small-scale model and an actual room/building.

The multizone, zonal, and CFD models are numerical models. The multizone models predict ventilation performance in a whole building. These models could only provide average characteristics of airflows and contaminant distributions due to the various assumptions used. For a whole building, the multizone models are the best choice. The zonal models could provide more information for flow in a zone (room). However, these models are rather difficult to use. Success in using zonal models often requires prior knowledge of airflow in a zone, which may not be realistic. The CFD models provided the most detailed information about the performance of ventilation systems in a zone, and were the most accurate of the numerical models, but they are sophisticated and require very dedicated training of a user. Although it is possible to apply the CFD models to a whole building, the computing time would be extremely long – too long for design. Since the CFD models used approximations, it is essential to validate the models. A validated CFD model is a powerful tool for conducting a parametric study or for optimizing the design of ventilation systems in buildings.

The above conclusions indicate that no universal model is available for predicting ventilation performance. Selection of an appropriate model depends on the problem studied.

Acknowledgements

The authors wish to thank Dr. Yi Lin and Mr. Brent Griffith for their contributions to the zonal model calculations. This study was funded partially by the U.S. Federal Aviation Administration (FAA) Office of Aerospace Medicine through the National Air Transportation Center of Excellence for Research in the Intermodal Transport Environment under Cooperative Agreement 07-C-RITE-PU, and partially by China's Ministry of Education through its Chang Jiang Scholars Program. Although the FAA has sponsored this project, it neither endorses nor rejects the findings of this research. The presentation of this information is in the interest of invoking comments from the technical community on the results and conclusions of the research.

References

- [1] Chen Q. Ventilation performance prediction for buildings: a method overview and recent applications. *Building and Environment*. 2009;44(4):848-858.
- [2] Mazumdar S, Chen Q. A one-dimensional analytical model for airborne contaminant transport in airliner cabins. *Indoor Air*. 2008;DOI: 10.1111/j.1600-0668.2008.00553.x
- [3] Powers DL. *Boundary value problems and partial differential equation*. Elsevier Academic Press. 2006
- [4] Lee KS, Jiang Z, Chen Q. Air distribution effectiveness with stratified air distribution systems (RP-1373). *ASHRAE Transactions*. 2009;115(2).
- [5] Poussou SB. Experimental investigation of airborne contaminant transport by a human wake moving in a ventilated aircraft cabin. Ph.D Thesis, Purdue University, IN. 2008
- [6] Karasso PS, Mungal MG. PLIF measurements in aqueous flows using the Nd:Yag laser. *Exp. Fluids*. 1997;23:382-387.
- [7] Zhang Y, Sun Y, Wang A, Topmiller J, Bennett J. Experimental characterization of airflows in aircraft cabins, Part 2: Results and research recommendations. *ASHRAE Transactions*. 2005;111(2): 53-59.

- [8] Zhang Z, Zhang W, Zhai Z, Chen Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-2: comparison with experimental data from literature. *HVAC&R Research*. 2007;13(6):871-886.
- [9] Walton GN, Dols WS. *CONTAMW 2.4 user manual*. National Institute of Standards and Technology, Gaithersburg, MD. 2006.
- [10] Wang L, Chen Q. Applications of a coupled multizone and CFD model to calculate airflow and contaminant dispersion in built environment for emergency management. *HVAC&R Research*. 2008;14(6):925-939.
- [11] Haghighat F. and Megri AC. A comprehensive validation of two air flow models: COMIS and CONTAM. *Indoor Air*, 1996;6:278-288.
- [12] Wang, L. and Chen, Q. Validation of a coupled multizone and CFD program for building airflow and contaminant transport simulations. *HVAC&R Research*, 2007;13(2):267-281.
- [13] Haghighat F, Lin Y, Megri, AC. Development and validation of a zonal model-POMA. *Building and Environment*. 2001;36:1039-1047.
- [14] Inard C, Bouia H, Dalicieux P. Prediction of air temperature distribution in buildings with a zonal model. *Energy and Buildings*. 1996;24:125-132.
- [15] Huang H. and Haghighat F. An integrated zonal model for predicting indoor airflow, temperature, and VOC distributions. *ASHRAE Transactions*, 2005;111(Pt 1), 601-611.
- [16] Zhang Z, Zhang W, Zhai J, Chen Q. Evaluation of various CFD models in predicting room airflow and turbulence. *HVAC&R Research*. 2007;13(6):853-870.
- [17] Yakhot V, Orszag SA. 1986. Renormalization group analysis of turbulence. *Journal of Scientific Computing*. 1986;1:3-51.
- [18] Yuan X, Chen Q, Glicksman LR, Hu Y, Yang X. Measurements and computations of room airflow with displacement ventilation. *ASHRAE Transactions*. 1999;105(1):340-352.