Chen, Q. 2009. "Ventilation performance prediction for buildings: A method overview and recent applications," *Building and Environment,* 44(4), 848-858.

# Ventilation performance prediction for buildings: A method overview and recent applications

Qingyan Chen[a,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

This paper presented an overview of the tools used to predict ventilation performance in buildings. The tools reviewed were analytical models, empirical models, small-scale experimental models, full-scale experimental models, multizone network models, zonal models, and Computational Fluid Dynamics (CFD) models. This review found that the analytical and empirical models had made minimal contributions to the research literature in the past year. The small- and full-scale experimental models were mainly used to generate data to validate numerical models. The multizone models were improving, and they were the main tool for predicting ventilation performance in an entire building. The zonal models had limited applications and could be replaced by the coarse-grid CFD models. The CFD models were most popular and contributed to 70 percent of the literature found in this review. Considerable efforts were still made to seek more reliable and accurate models. It has been a trend to improve their performance by coupling CFD with other building simulation models. The applications of CFD models were mainly for studying indoor air quality, natural ventilation, and stratified ventilation as they were difficult to be predicted by other models.

*Keywords:* Analytical; Empirical; Small scale; Full scale; Environmental measurements; Multizone; Zonal; Computational fluid dynamics (CFD); Numerical simulations

## 1. Introduction

Ventilation is used in buildings to create thermally comfortable environment with acceptable indoor air quality by regulating indoor air parameters, such as air temperature, relative humidity, air speed, and chemical species concentrations in the air. In order to regulate the indoor air parameters, it is essential to have suitable tools to predict ventilation performance in buildings. Ventilation performance prediction is to provide the information concerning indoor air parameters in a room or a building even before the building is constructed. Very often, one can assume that the air in a room is well mixed, which implies uniform distributions of air temperature and chemical species concentrations. For small rooms, such as small offices, hotel rooms, and bedrooms, such an assumption is often acceptable. For large spaces, like theaters, hotel lobbies, and gymnasia, the complete mixing assumption may not be acceptable. One would need the distributions of air temperature, relative humidity, air speed, and chemical species concentrations to assess ventilation performance.

Ventilation performance can be typically predicted or evaluated by analytical and empirical solutions, experimental measurements, and computer simulations. This paper is to provide an overview of those methods and some of their recent applications for predicting ventilation performance in buildings.

## 2. Overview of methods for predicting ventilation performance and their recent applications

This paper intended to give an overview of the most popular methods for predicting or evaluating ventilation performance, such as analytical models, empirical models, small-scale experimental models, full-scale experimental models, multizone models, zonal models, and Computational Fluid Dynamics (CFD) models. There were hybrid models of two or more different types. Majority of the hybrid models are coupled with the CFD models and will be discussed together with the CFD models. Hundreds, if not thousands, recent applications of these analytical, empirical, experimental, and computational models can be found in the literature for predicting ventilation performance in buildings. We have studied the publications in the past three years in major English language journals. This paper, however, only used these published in the past year that were by no means inclusive but an indication of the trend in predicting ventilation performance in buildings.

### *2.1 Analytical models*

Analytical models are derived from fundamental equations of fluid dynamics and heat transfer, such as mass, momentum, energy, and chemical-species conservation equations. The analytical models use simplifications in both geometry and thermo-fluid boundary conditions in order to obtain a solution. As a result, the final equations obtained for one case may not be used for another without modifications. However, the methodology and approximations could be similar for difference cases.

An example of the analytical models is the one developed by Fitzgerald and Woods [1] who studied the influence of stacks on flow patterns and stratification associated with natural ventilation with two openings as shown in Fig. 1. The analytical model calculates the temperature elevation in the room with a distributed heat flux, $Q_H$, by

$$\Delta T = \left( \frac{Q_H^2}{\alpha \rho^2 C_p^2 A^{*2} g(H - h_B)} \right)^{1/3} \tag{1}$$

and the flow rate by

$$V = \left( \frac{A^{*2}(H - h_B) g \alpha Q_H}{\rho C_p} \right)^{1/3} \tag{2}$$

where $A^*$ is the effective area of the two openings.

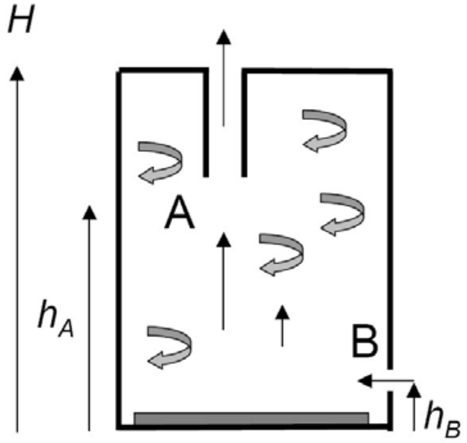


*Fig. 1. Schematic of the steady ventilation regimes in a room heated by a distributed source at the base and ventilated by two openings use by Fitzgerald and Woods [1]*

Another example of analytical models is the one developed by Mazumdar and Chen [2] using the principle of superposition and the method of separation of variables. They obtained an analytical solution of contaminant concentration, $C$, in an airline cabin as a function of position x for a contaminant source located at $L_1$ to be:

$$C = C_{inlet} + A_{L_1} e^{m_1(L_1 - x)} + B_{L_1} e^{m_2(L_1 - x)} + a_0 e^{-\beta_0^2 t} + 2 \sum_{n=1}^{\infty} a_n \cos\left[\alpha_n \left(L_1 - x\right)\right] e^{-\beta_n^2 t} \tag{3}$$

where $C_{inlet}$ is the contaminant concentration from the air supply inlet and the coefficients ($A, B, a, \alpha, \beta$) can be obtained from mathematical equations with a lot of approximations.

The analytical models are probably the oldest method for predicting ventilation performance. This method is still widely used today due to its simplicity, rich in physical meaning, and little requirement in computing resources, although it may not be accurate for complicated ventilation cases and the results may not be informative.

The following examples of the analytical method are a few recent applications. Holford and Woods [3] used analytical models to study thermal buffering of naturally ventilated buildings through internal thermal mass. They found that the role of thermal mass in buffering the interior temperature was very different under different ventilation rates. By analyzing natural ventilation with solar chimney, Bassiouny and Koura [4] obtained a simple relationship between solar intensity and room air temperature. Applying an analytical method involving a sinusoidal pulse of outdoor concentration to estimate indoor contaminant concentrations caused by outdoor contaminants, Halios and Helmis [5] predicted the time lag and the reduction of the indoor contaminant fluctuations. Coffey and Hunt [6] developed different analytical models of calculating ventilation effectiveness to evaluate mixing and displacement ventilation. It is interesting to note that analytical models sometime can be used to assess whether a more advanced model would work. For example, Wu et al. [7] used an analytical solution to assess an airflow network model for calculating air temperature and flow rate for complicated ventilation systems.

These results indicate that the analytical method is still very useful nowadays. The method is a powerful tool for predicting ventilation performance. The analytical models can give qualitative and sometime quantitative indication of the influence on ventilation performance caused by different geometry and thermo-fluid boundary conditions. The contributions of the analytical models to the research literature are minimal as it has been developed for decades.

### *2.2 Empirical models*

Similar to the analytical models, the empirical models are developed from the conservation equations of mass, energy, and chemical species. In many cases, the data of experimental measurements or advanced computer simulations are also used in the development of the empirical models to obtain some coefficients that make empirical models work in a certain scope. In theory, the analytical and empirical models do not differ very much. The perception is that the empirical models may use more approximations than the analytical models.

Typical empirical models for predicting ventilation performance are jet formulae that calculate air velocity, air temperature, and chemical-species concentration profiles. The formulae have been a bread and butter tool for design engineers to estimate thermal comfort and indoor air quality in a ventilated space. For example, by using CFD and experimental results of wall confluent jets in a room, Cho et al. [8] developed a set of equations to determine the jet behavior in terms of velocity profiles, the spreading rate of jets on the surface, and jets decay. For wall confluent jets, they calculated the maximum velocity, $U_m$, as

$$\left(\frac{U_m}{U_o}\right) = 2.96 l_c^{-0.79} \tag{4}$$

where $U_o$ is supply velocity of the jet and $l_c$ is characteristic length. Note that the throw constant (2.96) was empirically obtained. Such empirical formulae can be found in most design handbooks and design guides. It is a symbol of maturity in engineering practice.

There are thousands of empirical models for different ventilation performance assessment. The following is a few recent applications. Cornick and Kumaran [9] examined four popular empirical models for predicting interior relative humidity by comparing them with the measured data. NIOSH [10] used the data from 67 airborne infection isolation rooms to develop an empirical model describing the relationship between flow rate, pressure differential and leakage area. The model could effectively estimate the actual leakage area in these rooms. By developing a model for cross-ventilated buildings with two large external openings, Bastide et al. [11] obtained a building-dependent coefficient that better represented the turbulent phenomena near large external openings. Mahdavi and Pröglhöf [12] used an empirical model calibrated with in-situ data to study control of natural ventilation in buildings. The empirical models can also be combined with analytical models to provide more information. For example, Nazaroff [13] used analytical models and empirical data to study intake fraction for episodic indoor pollutant releases, such as those from cleaning, cooking, or smoking. The results indicated that the intake fraction in general depended on building-related, occupant, and pollutant dynamic factors, but a simplified relationship could be established for simple cases.

These applications of the empirical models demonstrate that the models are effective, cost-cutting tools for ventilation engineers and designers to predict ventilation performance in buildings. The performance of the empirical models is similar to that of the analytical models. They are very case dependent. The references collected over the past three years show that the empirical models did not contribute much to the research literature, after a few decades of development.

### *2.3 Small-scale experimental models*

The small-scale experimental models use measuring techniques to predict or evaluate ventilation performance with a reduced scale of the buildings or rooms. It is much more economical to use a small-scale experimental model than a full-scale building or room. One can get realistic ventilation performance by directly measuring thermo-fluid conditions in a small-scale model if the flow in the model is similar to these in reality. In order to achieve flow similarity between a small-scale experimental model and a real building or room, important dimensionless flow parameters in a small-scale experimental model such as Reynolds number, Grashof number, Prandtl number, etc. must remain the same as those in the actual building or room. When heat transfer is involved in a room with ventilation, it is difficult to obtain the same Reynolds and Grashof numbers. One possibility is to use liquid with different density, such as water or Freon to simulate thermal buoyancy. Otherwise, the small-scale model may not simulate the actual flow in buildings or rooms. Even though, the flow in the small-scale may not be the same as that in the actual room.

Livermore and Woods [14] used a small acrylic tank of 31 cm height filled with water, as shown in Fig. 2, to study natural ventilation of a building with heating at multiple levels. The building had two floors connected to a common atrium on the left. Both floors were heated and contained a series of openings to represent natural ventilation areas. The figure showed the flow direction through the openings and within the tank using the shadow graph technique and enhanced by tracer dyes. They also developed analytical models to calculate the height of neutral buoyancy and the flow direction. The results from the analytical model and the small-scale model were in close agreement. The group used the same technique to study stratification and oscillations produced by pre-cooling during transient natural ventilation in a building [15].

*Fig. 2. Photograph of a small-scale model filled with water for flow visualization of a two-storey building with a common atrium (Livermore and Woods 2007)*

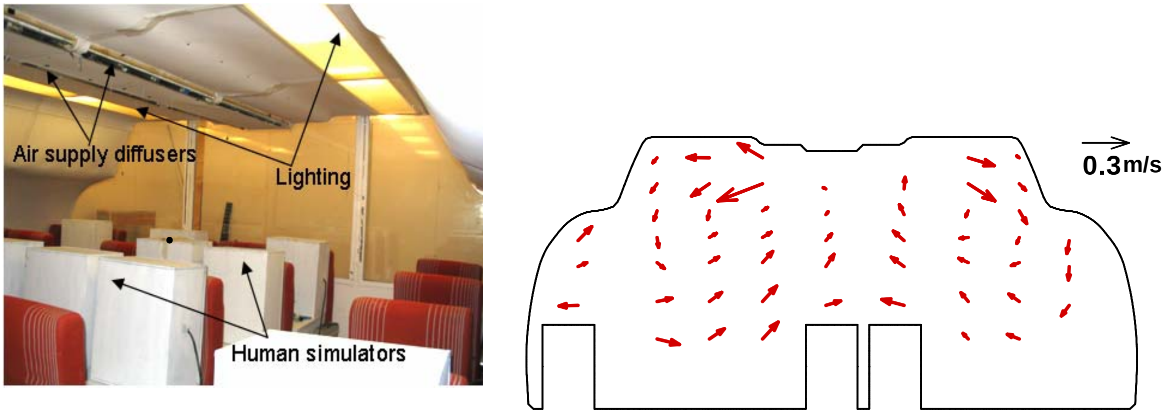
Yu et al. [16] studied airflow in a ceiling slot-ventilated enclosure in a 1:3 scale model. They measured airflow pattern, centerline velocity and temperature decays, velocity and temperature profile, airflow and thermal boundary layers, etc. The data were used to develop empirical models. With several 1:10 scale models, Morsing et al. [17] determined the effects of internal airflow and floor design on gaseous emissions from animal houses. Tapsoba et al. [18] compared the airflow patterns in a slot-ventilated enclosure partially loaded with empty slotted boxes in a 1:3.325 scale model measured by a laser Doppler anemometer and computed by CFD. The measured flow patterns agreed with the computed results. Similar approach was used by Lee et al. [19] to validate a CFD model for development of a new ventilation system. Kang and Lee [20] used a scale model to study natural ventilation in a large factory building using a louver ventilator.

The small-scale experimental models are very effective and economical to study ventilation performance in buildings. However, in addition to scaling issues associated with thermo-fluid dimensionless parameters, it can be rather challenging to scale down complex flow geometry, for example a complex diffuser. Our literature review found that instead of being used directly to study ventilation performance, the small-scale experimental models were mainly used to validate analytical, empirical, or numerical models. The validated analytical, empirical, or numerical models were then scaled up for studying the ventilation performance in real buildings.

## *2.4 Full-scale experimental models*

The full-scale experimental models have been widely used for predicting ventilation performance in buildings. However, our literature review found that the trend in using the full-scale experimental models was similar to that in using small-scale experimental model. The full-scale models were mainly used to generate data to validate numerical models, especially CFD models. Many well-known experimentists started to switch their career to use numerical models.

The full-scale experimental models can be further classified into two categories: laboratory experiment and in-situ measurements. Laboratory experiment often uses an environmental chamber to mimic a room or a single storey building with several small rooms. If outdoor wind conditions have to be considered, the chamber should be placed in a wind tunnel, which would make the facility very expensive. In a chamber, the thermo-fluid boundary conditions can usually be controlled. Zhang et al. [21] used an environmental chamber to mimic a section of a twin-aisle airliner cabin shown in Fig. 3(a). Even a full-scale experimental model often approximated thermo-fluid boundary conditions and flow geometry. For instance, Zhang et al. simulated passengers by heated boxes, as it could be unethical to employ many people as "passengers" in the cabin mockup where tracer gas and particles needed to be released for contaminant transport simulation. Nevertheless, the experimental facility looked rather realistic. One can use suitable equipment to measure ventilation performance. For instance, Zhang et al. used ultrasonic anemometers to measure the distributions of air velocity, air temperature distributions, and contaminants simulated by a tracer-gas and mono-size particles. The contaminants were assumed to be viruses released from a passenger with an infectious disease. Fig. 3(b) shows the air velocity measured in a cross section of the cabin.



*(a) A section of airliner cabin* *(b) Air velocity distribution measured at a cross section*

*Fig. 3. Experimental measurements in a section of a full-scale, twin-aisle airliner cabin used by Zhang et al.[21]*

It may not be realistic to construct a full-scale model, if one wants to predict ventilation performance in a theater or in an entire multi-storey building. One solution is to use an existing building of similar kind to predict the ventilation performance. Such in-situ measurements can be difficult because the thermo-fluid boundary conditions are not controllable in most cases. There may be unexpected disturbances during an experimental measurement. The resolution of data is often very low because it may not be practical to measure ventilation parameters in many locations in a large building. In addition, the data obtain from one building may not be applicable to a similar building nearby.

Please note that experimental measurements are not free from errors [22]. The measuring equipment needs frequent calibration and has its limitations. For example, a hot-wire or hot-sphere anemometer may not measure air velocity correctly when it is low. Sun and Zhang [23] gave an excellent overview on equipment available for measuring air velocity. Sandberg [24] also presented detailed information on using particle image velocimetry or particle streak velocimetry to measure entire air velocity field in a ventilated room.

In the past year, there were considerable amount of studies on predicting ventilation performance using full-scale models. Larsen and Heiselberg [25] used the data from a full-scale wind-tunnel experimental facility to establish a new expression for calculating the airflow rate in single-sided natural ventilation. In the same facility but for cross ventilation in a building, Nishizawa et al. [26] found that main flow, rebounding and changing flow direction, deflected flow, surface flow, and circulating flow were very important for cross ventilation. Wang et al. [27] used a section of a full-scale, twin-aisle aircraft cabin containing 35 mannequins to evaluate the ventilation effectiveness.

Many in-situ measurements on ventilation performance were found in the literature in the past year. Stathopoulou et al. [28] measured air quality in two large athletic halls with natural and mechanical ventilation in relation to outdoor pollution and meteorological conditions. By measuring indoor temperature and humidity in four traditional Japanese buildings, Yoshino [29] concluded that the traditional cooling technologies such as solar shading by thatched roof could decrease indoor temperature. Pajumägia et al. [30] conducted air temperature and relative humidity measurements in 11 cowsheds and found that the spatial temperature distribution was a measure to assess the ventilation efficiency. With the measurements of temperature and $CO_2$ concentration in five mechanically and four naturally ventilated office buildings, Hummelgaard et al. [31] correlated the symptoms and adverse perceptions of the occupants with ventilation. By measuring ventilation in a house with a double skin façade system, Xu and Ojima [32] could quantify energy demand reduced by the system.

The recent applications indicate that the full-scale models by laboratory experiment or in-situ measurements give the most realistic prediction of ventilation performance for buildings. However, they were generally very expensive and time consuming. In addition, these experimental measurements were not free from errors. Current trend seems to use full-scale experimental models of laboratory experiment and in-situ measurements to obtain data for validating computer models, such as CFD models, and then use the validated computer models to conduct the predictions of ventilation performance or design ventilation systems. The in-situ measurements were more frequently used to evaluate the performance of existing buildings.

### *2.5 Multizone models*

The multizone network models are mainly used to predict air exchange rates and airflow distributions in buildings with or without mechanical ventilation systems. They can also be used to calculate ventilation efficiency, energy demand, pollutant transport, and smoke control. A comprehensive background and theory of multizone models can be found in reference [33]. The multizone models solve mass, energy and chemical-species conservation equations. However, the models assume quiescent or still air in a zone so that the momentum effect can be neglected. The models further assume uniform air temperature, and chemical-species concentration in a zone. Wang and Chen [34] found that the assumptions could cause significant errors in some cases. They proposed to solve the problem by coupling a multizone program, CONTAM, with a CFD program, and proved theoretically that the coupling had a solution and the solution was unique [35]. They [36] also validated the coupled the program with experimental data and found that the coupled program could significantly improve the results.

A typical multizone model calculates the airflow and contaminant transport 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 default positive direction of $F_{ij}$ is defined to be from zone $i$ to zone $j$. In a multizone model, $F_{ij}$ is often calculated by a power-law function of the pressure drop, $\Delta P_{ij}$, across path $ij$,

$$F_{ij} = \alpha_{ij} c_{ij} \, |\Delta P_{ij}|^{n_{ij}} \tag{5}$$

where, $c_{ij}$ is the flow coefficient, $n_{ij}$ is the flow exponent of path $ij$, $\alpha_{ij} = (P_i - P_j)/|P_i - P_j|$, $P_i$ and $P_j$ are the total pressure at either zone i or j side of path $ij$, respectively. The multizone model solves air mass balance equations under steady-state conditions for zone $j$ as

$$\sum_i F_{ij} + F_j = \sum_i \alpha_{ij} c_{ij} \, |\Delta P_{ij}|^{n_{ij}} + F_j = 0 \tag{6}$$

where, $F_j$ is the air mass sources in zone $j$. Similarly, contaminant or chemical-species mass balance at steady-state condition in zone $j$ is,

$$\sum_i F_{ij} C_{IJ} + S_j = 0 \tag{7}$$

where, $C_{IJ} = C_i$ when the airflow is from zone $i$ to zone $j$ ($F_{ij} > 0$), and $C_{IJ} = C_j$ when the airflow is from zone $j$ to zone $i$ ($F_{ij} = -F_{ji} < 0$); $C_i$ and $C_j$ are the contaminant concentrations in zone $i$ and zone $j$, respectively; $F_{ij}$ is the airflow rate from zone $i$ to zone $j$; and $S_j$ is the contaminant sources inside zone $j$. Fig. 4 shows an example of airflow calculated by CONTAM in many zones on the second floor of a three-storey building. The length of the line indicated the magnitude of the flow rate. The figure depicted that the air came from west to the building and left the building through the other three façades. There were flows between different indoor spaces (zones).

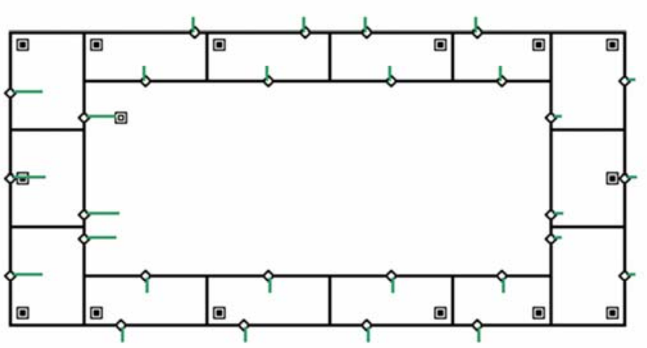


*Fig. 4. Airflow in a building calculated by a multizone model, CONTAM*

Hu et al. [37] employed CONTAM to calculate particle resuspension in the indoor environment of a three-zone building, and found that CONTAM could reach a very fast convergence speed and the results agreed with those from the analytical model. They also tried to tune the model by changing airflows, flow resistances, and other parameters [38]. Wang and Chen [39] coupled CONTAM with a CFD program to study the effectiveness of using emergency ventilation to protect the building occupants in case of a toxic gas release. CONTAM was also applied to calculate indoor air quality for a building using indoor air cleaners [40].

COMIS is another popular multizone program. It was employed to calculate the effect of the wind speed velocity on the stack pressure in a building [41], to predict airflow, pressure, and contaminant distribution in a building [42], and to determine airflows between zones due to temperature differences [43]. The performance of CONTAM and COMIS seems similar. As discussed by Crawley et al. [44] and Megri [45], multizone models had also been combined with energy analysis or load calculation programs. Although multizone models were not very accurate in each zone due to the assumptions used, they were very powerful design tools, especially for calculating airflow in a large building. The accuracy in each zone could be remedied with more detailed airflow program, such as a zonal model or a CFD model [36].

Unlike most commercial CFD programs, the two popular multizone models CONTAM and COMIS were developed by two national laboratories in the United States. They do not have very user-friendly interface for data input, and the graphical presentation of the results are not attractive. These deficiencies may have severely limited the applications of the multizone models in practice. According to our private communications with many designers, the multizone models seem to be the only tool to obtain meaningful results for predicting ventilation performance in an entire building.

### *2.6 Zonal models*

The well-mixing assumption used in the multizone models is not valid for large indoor spaces or a room with stratified ventilation system, such as displacement ventilation. Therefore, zonal models have been used to remedy the problem in predicting the distributions of air temperature. Zonal models divide a room into a limited number of cells, typically less than 1000 for a three-dimensional space. Air temperature is calculated in each cell to determine its non-uniform distribution in the space.

Mergi and Haghighat [46] reviewed the development of the zonal models. The zonal models had been developed based on measured airflow patterns or mass and energy balance equations. Those based on measured airflow patterns relied on the patterns to calculate air temperature distributions. Their applications were limited by the availability of airflow patterns. The models used mass and energy balance equations were the mainstream ones at present. The mass balance equation can be written as

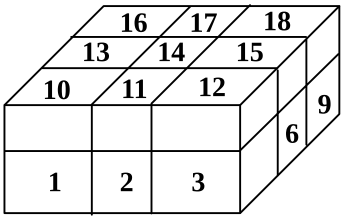
$$\sum_j \dot{m}_{j\to i} = 0 \tag{8}$$

where $\dot{m}_{j\to i}$ is the mass flow rate from neighboring cell, *j*, to current cell, *i*. The energy balance equation is

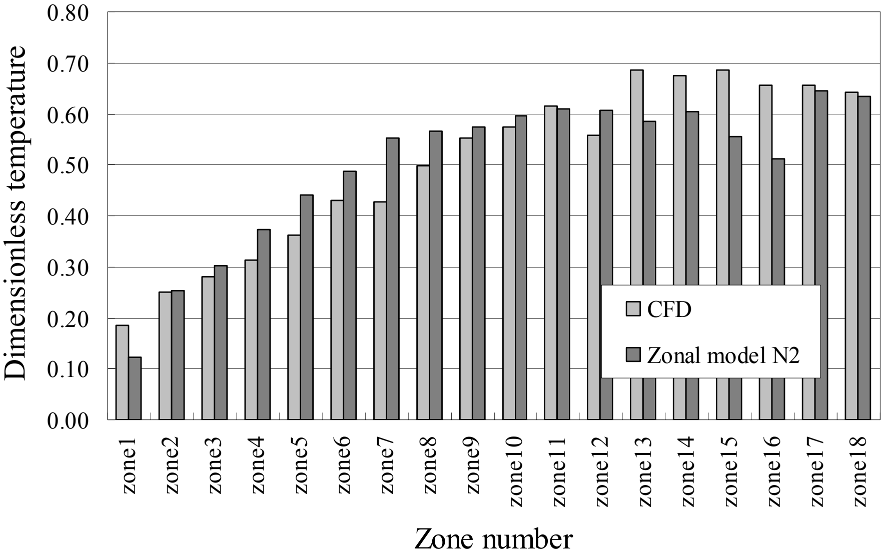
$$\sum_j \Phi_{j\to i} + \Phi_{source} = \rho_i V_i c_p \frac{\partial T_i}{\partial t} \tag{9}$$

where $\Phi_{j\to i}$ is the energy flow rate from neighboring cell, $j$, to current cell, $i$; $\Phi_{source}$ is the heat source from cell $i$; and the term in the right side of the equation is the energy accumulated in cell $i$.

Song et al. [47] developed a zonal model by further integrating it with dynamic models for heat and moisture transfer and source/sink models for air pollutants. They applied the model to calculate the dynamic air temperature, humidity, and pollutant concentrations in a room with displacement ventilation. Fig. 5(a) shows 18 zones used in their zonal model and Fig. 5(b) compares the air temperatures at these zones with the corresponding CFD results. The performance of the zonal model is pretty good.



*(a) The 18 zones used for the room*



(*b) Comparison of calculated temperatures by the zonal model and by CFD in each zone*
*Fig. 5. An example of results obtained by using a zonal model by Song et al. [47]*

Recent applications include the integration of the zonal model with other models to calculate not only indoor temperature, humidity, and pollutant concentrations, but also the heating/cooling load of HVAC systems [48]. Jiru and Haghighat [49] used a zonal model for modeling airflow and temperature in a ventilated double-skin façade system.

Zonal models have also been used to predict climate dynamics in ventilated bulk-storage of agricultural produce [50].

Note that most applications in the past year were for flow with weak momentum forces in the room air. If the flow momentum were strong, the accuracy of the zonal model simulations would suffer considerably. This is because the zonal models based on mass and energy balance equations do not solve momentum equation in order to reduce computing costs. In the jet region or thermal plume region where the momentum is strong, special treatments are needed that would increase significantly the complicities of the zonal models. The complicities would also increase the computing costs and make the equation system in the zonal models less stable. In this case, a modified zonal model may require similar computing effort as CFD with the same number of cells. Zonal models based on measured airflow patterns provide little incentive to calculate temperature distribution, because it may be more direct just to measure the temperature with a minimal additional effort.

The literature shows that most of studies on zonal models concern the development of zonal models. Only a handful of them were regarding practical applications. The use of zonal model is not as easy as one may have thought; especially one has to handle special cells. By comparing with a very coarse-grid CFD simulations, the zonal models do not show much superiority on reducing computing time. In many cases, the overhead time in preparing data input for a zonal model may be longer than that for a CFD simulation. In the future, the zonal models could be replaced by the CFD models as computers become even faster and CFD interface is more user-friendly.

### *2.7 CFD models*

The Computational Fluid Dynamics (CFD) numerically solves a set of partial differential equations for the conservation of mass, momentum (Navier-Stokes equations), energy, chemical-species concentrations, and turbulence quantities. The solution provides the field distributions of air pressure, air velocity, air temperature, the concentrations of water vapor (relative humidity) and contaminants, and turbulence parameters for both indoor and outdoor spaces. Despite having some uncertainties in the models, requiring sufficient knowledge on fluid mechanics from a user and demanding a high capacity computer, the CFD models become more and more popular in predicting ventilation performance due to the rapid increase in computer capacity and the development of user-friendly CFD program interfaces.

The CFD models have been widely used to study indoor air quality, thermal comfort, fire safety, HVAC system performance, etc. in various buildings (commercial buildings, residential buildings, schools, health care facilities, institutional buildings, and industrial buildings), underground facilities, public transportation vehicles, greenhouses, animal facilities, etc. Norton [51][52] conducted very comprehensive reviews of the CFD applications for ventilation studies in food and agricultural industry (greenhouses and animal production facilities). The CFD applications in these two sectors were not discussed in this paper. To make this review manageable, the CFD applications in fire safety studies in buildings were further excluded. Although the principle of CFD for ventilation studies is quite similar to that for fire safety investigations, the time scale concerning a problem and temperature variation differ substantially. Thus, this review focused on the area of predicting general ventilation performance in buildings for human beings. Fig. 6 shows the numbers of papers found in major journals on CFD applications for predicting ventilation performance. The figure depicts that the applications of CFD are mushrooming.

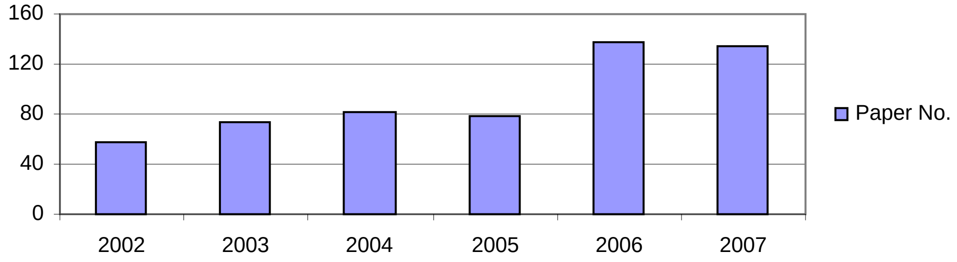


*Fig. 6. Paper numbers published from 2002 to 2007 in major journals using CFD to predict ventilation performance in buildings occupied by human beings*

The CFD models generally include Reynolds Averaged Navier-Stokes equation (RANS) modeling and Large Eddy Simulation (LES). The RANS modeling solves a set of transport conservation equations for continuity, momentum, energy, and chemical-species concentrations. For an incompressible Newtonian flow, the Navier-Stokes equation can be written as

$$\frac{\partial U_i}{\partial t}+\frac{\partial}{\partial x_j}\left(U_j U_i\right)=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\frac{\mu}{\rho}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\overline{u_i u_j}\right)+S \tag{10}$$

where the bar stands for Reynolds average. The turbulence Reynolds stresses, $\overline{u_i u_j}$, can be solved directly by an additional set of transport conservation equations. The approach is called Reynolds stress modeling. The other approach is to link the turbulence Reynolds stresses to Boussinesq eddy-viscosity approximation as

$$-\overline{u_i u_j}=\frac{\mu_t}{\rho}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}k \tag{11}$$

where $\mu_t$ is an eddy viscosity that should be determined. This approach is known as eddy viscosity models. Depending on how many transport equations are used to determine the eddy viscosity, the eddy viscosity models are further categorized as zero-, one-, two-, three-, and four-equation models. The most well known two-equation models are the standard k-ε model [53] and the RNG k-ε model [54].

The LES requires the separation of small-eddies from large-eddies with a filter. For simplicity, the following section uses a one-dimensional notation. The filtered velocity is

$$\overline{U_i}=\int G(x,x')U_i(x)dx' \tag{12}$$

where *G(x, x')* is a filter function. The filter function is large only when *G (x, x')* is less than the filter width, a length scale over which the averaging is performed. The eddies larger than the filter width are “large-eddies”, and those smaller than the width are “small-eddies”. If a box filter is used, i.e.

$$G(x_i) = \begin{cases} \frac{1}{\Delta_i} & (|x_i| \le \frac{\Delta_i}{2}) \\ 0 & (|x_i| > \frac{\Delta_i}{2}) \end{cases} \tag{13}$$

it is possible to derive the governing conservation equations for momentum (Navier-Stokes equations), mass continuity, energy and chemical-species concentrations. The filtered Navier-Stokes equation for an incompressible Newtonian flow is:

$$\frac{\partial \overline{U_i}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{U_j U_i}) = -\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\frac{\mu}{\rho}\left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i}\right) - \tau_{ij}\right) + S \tag{14}$$

where the bar stands for filtered value that is different from Reynolds average. The subgrid Reynolds stresses are

$$\tau_{ij} = \overline{U_i U_j} - \overline{U_i} \cdot \overline{U_j} \tag{15}$$

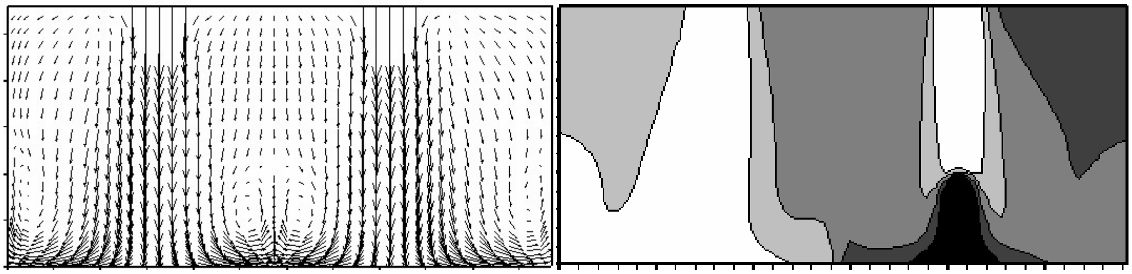
The Smagorinsky model [55] models the unknown $\tau_{ij}$ as

$$\tau_{ij} = 2C\overline{\Delta}^2 |\overline{S}| \overline{S}_{ij} \tag{16}$$

where $|\overline{S}| = (2\overline{S_{ij}} \cdot \overline{S_{ij}})^{\frac{1}{2}}$, $\overline{S_{ij}} = \frac{1}{2}(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i})$, $C = C_s^2$, and $C_s = 0.1 \sim 0.25$.

LES has only one or no empirical coefficient; thus it is superior to the RANS models. However one has to solve transient flow even the actual flow is steady and the flow details are not needed. The accuracy of an LES simulation depends on the grid resolution. Therefore, LES always requires much more computing time (at least two orders of magnitude longer) than RANS modeling for a steady-state flow.

Fig. 7 shows an example of CFD results obtained by Zhang and Chen [56] using a RANS model for a clean room with a contaminant source placed under the right diffuser and roughly one meter above the floor. Fig. 7(a) depicts the velocity vectors in a section through the two ceiling diffusers and Fig. 7(b) the contaminant concentration distribution in the section.



*(a) Velocity vector distribution* *(b) Contaminant concentration distribution*

*Fig. 7. The airflow and contaminant distribution in a section through the two diffusers in a cleanroom obtained by using a RANS model (Zhang and Chen 2007)*

The following subsections will present our review of CFD applications for predicting ventilation performance in human-occupied buildings in the past year. We actually went through the publications in the past three years and found the comments expressed in the following section to be generally valid. Due to space limit for a paper, this review only selected the papers with experimental validation or with other significance, such as new concepts.

### *2.7.1 CFD model validation*

At present all the CFD models use approximations for predicting ventilation performance for buildings. The approximations inevitable bring some uncertainties in predicting the distributions of airflow, air velocity, air temperature, and chemical-species concentrations. Even CFD has been applied to ventilation studies for over thirty years; engineers are still seeking for more accurate, more reliable, and faster CFD models. Zhai et al. [57] reviewed popular turbulence models that could be used for indoor environment modeling. They identified the most promising models from eight different categories including eddy viscosity models, Reynolds stress models, LES, and detached eddy simulation. Detached eddy simulation is a hybrid model that use LES for main flow region and RANS for boundary layer flow. Zhang et al. [56] then further validated these eight models for force convection and mixed convection in ventilated spaces, natural convection with medium temperature gradient in a tall cavity, and natural convection with large temperature gradient in a model fire room. They found that the performance of the models was not always consistent for different flows. Among the RANS models tested, the v2f-dav and RNG $k$-$\varepsilon$ models had the best overall performance. Many other researchers [58][59][60][61][62] tested various RANS models for predicting ventilation performance in buildings. These studies also concluded that one model could perform well for one flow but poorly in another. The performance of the RNG $k$-$\varepsilon$ model was rather stable.

The LES and detached-eddy-simulation could do a good job with a sufficient fine grid resolution. The LES seems more popular for predicting particle distributions in ventilated spaces, because particle predictions need detailed turbulent flow information that is available in LES prediction. Several studies [63][64][65][66] obtained good agreement between the predicted and measured particle concentrations in ventilated spaces. By looking at the general view on LES from the papers in the last three years, the researchers were more satisfied with the results than a few years ago despite higher computing costs. Currently, the LES is mainly used as a research tool. Gaining its popularity in design applications would still take time.

In addition to model validation, efforts were made on improving numerical schemes used in CFD. For example, Zitzmann et al. [67] developed a freeze flow technique to reduce computing costs of CFD by an adaptive control method that automatically adjusted the lengths of the frozen and unfrozen flow periods in the solution procedure. Ng et al. [68] searched for suitable high-order differencing schemes to solve indoor airflow, and Shakeri et al. [69] used different numerical grid presentations of an occupant.

### *2.7.2 Coupling CFD models with other building simulation models*

CFD models have also been used to improve other building simulation tools so that ventilation performance can be more accurately predicted. This has become a trend in areas involving multi-scale flow and heat transfer [70]. For example, researchers coupled energy simulation with CFD to improve the accuracy in natural ventilation prediction with reduced computing effort [71][72][73] and to study the performance of a double skin façade system [74]. Wang and Chen [35] coupled CFD with a multizone airflow program to improve the prediction of airflow and contaminant in an entire building when the effect of air momentum was strong, the temperature gradient was large, and the contaminant concentration variation was great in a zone. By coupling CFD with genetic algorithms, Kim et al. [75] determined the optimal design strategies for indoor thermal environment. Gao et al. [76] combined CFD and a multi-nodal human body thermo-regulation model to investigate the pollutant exposure reduction and thermal comfort with personalized ventilation.

Reviewing the papers on this topic in the past three years, the trend is clear that the researchers were satisfied with the coupling. The coupling did increase significantly the complexity, a price the researchers seemed rather willing to pay.

### *2.7.3 Applications of CFD models for indoor air quality studies*

Applications of CFD models for predicting ventilation performance in buildings at present can be generally divided into three categories: for indoor air quality studies in spaces with non-uniform distributions of contaminant concentrations, for natural ventilation designs, and for investigations on stratified indoor environments.

Zhang et al. [77] and Zhao et al. [78] [79] used a CFD model to study particle transport in different environment with high health concerns. Similar applications can also be found for a dental clinic [80]. Some applications [81][82] were geared to the transport and dispersion of the expiratory droplets and droplet nuclei in indoor environments. Other studies [83][84] were to compare Eulerian and Lagrangian methods in predicting particle dispersion in an enclosed environment.

In addition to particulate matters, CFD has also been used to study indoor air quality caused by gaseous contaminants. Examples are to evaluate indoor air quality in a polyvinyl chloride chemical plant [85], to assess worker exposures when toxic airborne contaminants were released into the wake region of a mannequin [86], to study inhaled air quality in a personal air-conditioning environment [87], and to develop ventilation efficiency indices for indoor environments in urban domains [88].

### *2.7.4 Applications of CFD models for natural ventilation designs*

Design of natural ventilation is very challenging due to the change of wind speed and direction over time as well as significant impacts of surrounding buildings on the ventilation rates through building openings. The multizone network models are often incapable to deal with such complicated scenarios of airflow. Due to the increasing interests in using natural ventilation to reduce energy demand and to improve indoor air quality, the applications of CFD models for natural ventilation design are popular at present [89].

Indeed, Asfour and Gade [90] found that CFD was a good tool to predict natural ventilation and could be used to develop/calibrate flow coefficients used in multizone model. For buoyancy-driven natural ventilation flows in a single-storey space connected to an atrium, the predicted airflow patterns, temperature distribution, and ventilation flow rates by CFD agreed well with those by the analytical models and the experiment [91]. CFD was also applied to optimize natural ventilation device designs [92][93]. Other examples of applications include the study of natural ventilation induced by wind flow around a high-rise building with a refuge floor [94], the analysis and design of airflow through a dual airflow window [95], and the verification of empirical models for natural ventilation in large enclosure [96].

### *2.7.5 Applications of CFD models for investigations on stratified indoor environment*

Simple models, such as analytical and empirical models, have great uncertainties when being used for stratified environments, including indoor spaces with displacement ventilation and underfloor air distribution. Therefore, CFD models have been widely used in such indoor environments for the investigations of ventilation effectiveness, indoor air quality, thermal comfort, etc.

Lau and Chen [97] used CFD to investigate the impact of air change rate, number of diffusers, diffuser location, occupant location, furniture arrangement, partition location, and arrangement of exhausts on indoor air quality in spaces with underfloor displacement ventilation. The same approach was used to analyze the age of air and other indoor environment parameters in an airliner cabin with underfloor air distribution systems and personalized ventilation systems [98] and to predict Air Diffusion Performance Index (ADPI) in a displacement-ventilated office [99]. Karimipanah et al. [100] investigated air quality and thermal comfort in stratified environment created by two floor-level air supply systems in classrooms.

Please note that the applications of the CFD models for predicting ventilation performance were not limited to these three areas. They were just taken as examples to show the recent trend in using CFD for various indoor environments where detailed analyses of ventilation performance were crucial. To give a sense of the CFD applications to other environments, here are a few recent examples. Yuan and You [101] evaluated and optimized ventilation systems for subway side-platform stations using CFD simulations. Bin and Sekhar [102] studied an air supply device with induced ventilation to evaluate its ability to create a comfortable environment. The investigations on mixing degree in enclosed environments [103][104] and on optimal sensor locations and minimal sensor numbers for sensing contaminants in commercial airline cabins[105][106] were also conducted by CFD simulations.

## 3. Discussion

The analytical models can only be applied to very simple problems. They are good tools in understanding ventilation mechanism. The models may have been used in daily life, but most of the analyses may not be original to be published in journals. This is probably due to a few decades of development that makes it difficult to find original contributions to the literature. Fig. 8 shows that about 3% of the publications in the past year on predicting ventilation performance are related to the analytical models.

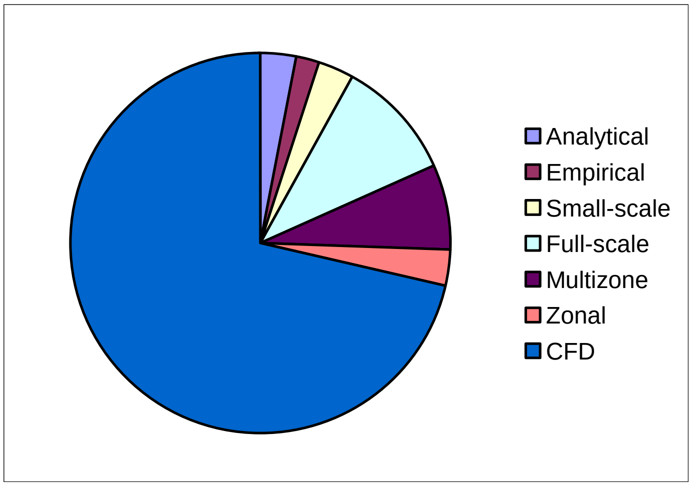


*Fig. 8. The share of different models in 2007 for predicting ventilation performance in buildings*

The empirical models are probably the bread and butter tools for ventilation design. Most of design handbooks, design guidelines, and product catalogs for designing ventilation in buildings use empirical models. About 2% of the publications on predicting ventilation performance in the past year used empirical models as illustrated in Fig. 8. The models are mainly for new ventilation systems, indicating the maturity of the method for conventional systems. Thus, the low appearance rate in the literature may not imply that they are not important.

The small-scale experimental models are less expensive than the full-scale experimental models. For this reason, the small-scale models still have their ground, despite of difficulties in modeling simultaneously inertial and buoyancy forces. The small-scale models often provide excellent flow visualization. By using particle image velocimetry, small-scale models can produce surprising high quality data for some types of flow, such as pure natural convection or pure forced convection. The small-scale models are hardly used alone as a predictive tool for ventilation performance. They are usually supplemented with analytical, empirical, and numerical models. At present, a significant number of studies have used small-scale models to produce quality data to validate more advanced computer models, such as CFD models. In the past year, 3% of the publications employ the small-scale models as a main tool for investigating ventilation performance. Counting the studies with both CFD and small-scale models, the share of the publications is about 5%.

The full-scale models in laboratory and in-situ measurements are very popular in ventilation performance studies. Most of the experimental measurements are again for generating data for CFD model validation, as they are very time consuming and expensive. High quality experimental data are still difficult to obtain due to the complexity of airflow in building ventilation. Many of the measurements do not provide detailed information, which makes it difficult for other researchers to use the data. This is especially true for in-situ measurements where experimentists may not even know the boundary conditions in their tests. Most people trust the data measured by experiment more than those computed by numerical simulations, but the trend shows that more and more experimentists are now using CFD models to supplement their experimental studies. The CFD modelers will continue to search for high quality experimental data to validate their simulations, although they can be rare. Fig. 8 depicts that about 10% of the publications on ventilation performance in the past year are based on full-scale experimental models.

The multizone network models are widely used in predicting ventilation performance for entire buildings. The models, after more than two-decade development, are reasonably solid. However, because of the heavy use of assumptions, serious efforts have been made to improve the models. The models are also improved with added capacities or by coupling them with other computer models. The multizone models will continue to be an important tool for designers. Currently, the interface for data input and graphical presentation of the results still lag behind, which could be the main obstacle for engineers to use the multizone models. In the past year, about 7% of the publications on predicting ventilation performance are related to the multizone models.

The zonal models are reduced order models that are intended for not-so-well-mixed air in buildings. Since the models do not solve the conservation equation for momentum, the accuracy suffers for the flow regions with strong momentum. Many remedy methods have been imposed, but they slow down the computation and increase the difficulty level in using the models. The literature in the past year shows some successful examples in more practical cases. However, a majority of the studies using zonal models are still for model development or for simple applications. The zonal models have still a long way to go to be a reliable and user-friendly tool for ventilation designers. Actually, the zonal models may be replaced by coarse-grid CFD simulations in the future, as the latter are not a lot of more computationally demanding. About 3% of the literature on ventilation performance studies in the past year used zonal models.

The CFD models accounts for 70% of the ventilation performance studies published in the past year as seen in Fig. 8, although this review narrows the CFD literature search down to a small area of predicting ventilation performance. It is interesting to note that about 2/3 of the studies used some kinds of experimental data to validate the CFD results, which clearly indicates that the community has yet trusted the CFD modeling without validation. That is why the researchers continue to seek for more reliable, more accurate, and faster CFD models. However, the effort has yet to produce fruitful results. Another trend is to couple the CFD models with other models to accomplish some impossible missions for a CFD model being applied alone. The works in these two areas account for 1/3 of the total CFD related publications on ventilation performance in the past year. The remaining 2/3 of the publications are mainly for studies concerning indoor air quality, natural ventilation, and stratified ventilation because the ventilation performance in these indoor spaces is difficult to be predicted with other methods. Most of the applications are practical and useful.

It is also interesting to note that LES has been used in quite a few ventilation studies, such as predicting particle trajectories. Clearly, these researchers put accuracy far above computing costs. However, this does not prevent the use of smart arrows with simple analytical equations for natural ventilation studies as the one used by Lomas [107].

In the foreseeable future, CFD will continue to be a research tool for predicting ventilation performance in buildings. The CFD models are becoming more and more popular in design practice. It is essential that the CFD models should be more reliable and faster. Reliability is a major issue at present since the validation by experimental data requires a lot of efforts regardless the data are obtained by the CFD modelers or from the literature. The CFD speed will continue to improve as computers are becoming faster. However, the demand on computing time will continue to increase as engineers would like to use more sophisticated CFD models to handle more complicated ventilation problems with even more grid cells. The CFD is indeed an attractive tool even with coarse cells used. The ASHRAE is currently funding a new research project 1418 "Optimizing the Trade Off between Grid Resolution and Simulation Accuracy: Coarse-Grid CFD Modeling".

This paper reviewed only the mainstream models used to predict ventilation performance in buildings. It is not inclusive. For example, the review did not include the state space model [108] and the probabilistic model [109]. In addition, the review was mainly for cause-effect problems, i.e. with known boundary conditions and system to determine the ventilation performance. The effect-cause or inverse problems in which one would try to find the cause with a known result were not covered in this review. The examples of inversing problem modeling related to ventilation performance can be found in references [110][111][112].

## 4. Concluding remarks

This paper presents an overview of the most popular methods for predicting ventilation performance, including the analytical models, empirical models, small-scale experimental models, full-scale experimental models, multizone models, zonal models, and CFD models. Reviewing the publications found in major journals, this paper revealed that the contributions from analytical and empirical models were around 5%, although they may be the bread and butter tools in practical design. Most of the studies conducted in the small-scale and full-scale experimental models were used for validation purpose. The multizone models were widely used for predicting ventilation performance in entire buildings. Serious effort has been made to improve the multizone models. Zonal models have yet to gain their popularity in predicting ventilation performance and may be replaced by coarse-grid CFD models in the future.

The CFD models were most popular for predicting ventilation performance in the research community in the past year. The researchers were seeking for more accurate and reliable CFD models. The trend of increasingly using LES deserves further attention. The use of CFD with other building simulation tools to enhance its ability and to reduce computing costs seems attractive. More such studies would expect to appear in the literature in the near future. For applied studies, there are many papers on indoor air quality investigations in spaces with non-uniform distributions of contaminant concentrations, natural ventilation designs, and investigations on stratified indoor environments. The CFD simulations have in fact been applied in many other related studies but they are beyond the scope of this review.

## Acknowledgements

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 technical community comment on the results and conclusions of the research.

## References

[1] Fitzgerald SD, Woods AW. The influence of stacks on flow patterns and stratification associated with natural ventilation. *Building and Environment* 2008; doi:10.1016/j.buildenv.2007.10.021.

[2] Mazumdar S, Chen Q. A one-dimensional analytical model for airborne contaminant transport in airliner cabins. Accepted by *Indoor Air*.

[3] Holford JM, Woods AW. On the thermal buffering of naturally ventilated buildings through internal thermal mass. *Journal of Fluid Mechanics* 2007;580:3-29.

[4] Bassiouny R, Koura NSA An analytical and numerical study of solar chimney use for room natural ventilation. *Energy and Buildings* 2008;40(5):865-873.

[5] Halios CH, Helmis CG. On the estimation of characteristic indoor air quality parameters using analytical and numerical methods. *Science of the Total Environment* 2007;381(1-3):222-232.

[6] Coffey CJ, Hunt GR. Ventilation effectiveness measures based on heat removal: Part 2. Application to natural ventilation flows. *Building and Environment* 2007;42(6):2249-2262.

**[7]** Wu Z, Melnik RVN, Borup F. Model-based analysis and simulation of airflow control systems of ventilation units in building environments. *Building and Environment* 2007;42(1):203-217.

[8] Cho Y, Awbi HB, Karimipanah T. Theoretical and experimental investigation of wall confluent jets ventilation and comparison with wall displacement ventilation. *Building and Environment* 2008;43(6):1091-1100.

[9] Cornick SM, Kumaran MK. A comparison of empirical indoor relative humidity models with measured data. *Journal of Building Physics* 2008;31(3):243-268.

[10] Hayden II CS, Earnest GS, Jensen PA. Development of an empirical model to aid in designing airborne infection isolation rooms. *Journal of Occupational and Environmental Hygiene* 2007;4(3):198-207.

[11] Bastide A, Allard F, Boyer H. Natural ventilation - A new method based on the Walton model applied to cross-ventilated buildings having two large external openings *International Journal of Ventilation* 2007;6(3):195-206.

[12] Mahdavi A, Pröglhöf C. A model-based approach to natural ventilation. *Building and Environment* 2008;43:620-627.

[13] Nazaroff WW. Inhalation intake fraction of pollutants from episodic indoor emissions. *Building and Environment* 2008;43(3):269-277.

[14] Livermore SR, Woods AW. Natural ventilation of a building with heating at multiple levels. *Building and Environment* 2007;42(3):1417-1430.

[15] Chenvidyakarn T, Woods A. Stratification and oscillations produced by pre-cooling during transient natural ventilation. *Building and Environment* 2007;42(1):99-112.

[16] Yu H, Liao C-M, Liang H-M, Chiang K-C Scale model study of airflow performance in a ceiling slot-ventilated enclosure: Non-isothermal condition. *Building and Environment* 2007;42(3):1142-1150.

[17] Morsing S, Strøm JS, Zhang G, Kai P. Scale model experiments to determine the effects of internal airflow and floor design on gaseous emissions from animal houses. *Biosystems Engineering* 2008;99(1):99-104.

[18] Tapsoba M, Moureh J, Flick D. Airflow patterns in a slot-ventilated enclosure partially loaded with empty slotted boxes. *International Journal of Heat and Fluid Flow* 2007;28(5):963-977.

[19] Lee I-B, Sase S, Sung S-H. Evaluation of CFD accuracy for the ventilation study of a naturally ventilated broiler house. *Japan Agricultural Research Quarterly* 2007;41(1):53-64.

[20] Kang J-H, Lee S-J. Improvement of natural ventilation in a large factory building using a louver ventilator. *Building and Environment* 2008;doi:10.1016/j.buildenv.2007.12.013.

[21] Zhang Z, Chen X, Mazumdar S, Zhang T, Chen Q. Experimental and numerical investigation of airflow and contaminant transport in an airliner cabin mockup. *Building and Environment* 2008;doi:10.1016/j.buildenv.2008.01.012.

[22] Melikov AK, Popiolek Z, Silva MCG, Care I, Sefker T. Accuracy limitations for low-velocity measurements and draft assessment in rooms. *HVAC&R Research* 2007;13(6):971-986.

[23] Sun Y, Zhang Y. An overview of room air motion measurement: technology and application. *HVAC&R Research* 2007;13(6):929-950.

[24] Sandberg M. Whole-field measuring methods in ventilated rooms. *HVAC&R Research* 2007;13(6):951-970.

[25] Larsen TS, Heiselberg P. Single-sided natural ventilation driven by wind pressure and temperature difference. *Energy and Buildings* 2008;40(6):1031-1040.

[26] Nishizawa S, Sawachi T, Narita K-I, Kiyota N, Seto H. Study of the airflow structure in cross-ventilated rooms based on a full-scale model experiment. *International Journal of Ventilation* 2007;6(1):51-59.

[27] Wang A, Zhang Y, Sun Y, Wang X. Experimental study of ventilation effectiveness and air velocity distribution in an aircraft cabin mockup. *Building and Environment* 2008;43(3):337-343.

[28] Stathopoulou OI, Assimakopoulos VD, Flocas HA, Helmis CG. An experimental study of air quality inside large athletic halls. *Building and Environment* 2008;43:834–848.

[29] Yoshino H. Passive cooling effect of traditional Japanese building's features. Management of Environmental Quality 2007;18(5):578-590.

[30] Pajumägia A, Poikalainen V, Veermäe I, Praks J. Spatial distribution of air temperature as a measure of ventilation efficiency in large uninsulated cowshed. *Building and Environment* 2008;43:1016-1022.

[31] Hummelgaard J, Juhl P, Sæbjörnsson KO, Clausen G, Toftum J, Langkilde G. Indoor air quality and occupant satisfaction in five mechanically and four naturally ventilated open-plan office buildings. *Building and Environment* 2007;42(12):4051-4058.

[32] Xu L, Ojima T. Field experiments on natural energy utilization in a residential house with a double skin façade system. *Building and Environment* 2007;42:2014–2023.

[33] Axley J. Multizone airflow modeling in buildings: History and theory. *HVAC&R Research* 2007;13(6):907-928.

[34] Wang L, Chen Q. Evaluation of some assumptions used in multizone airflow network models. *Building and Environment* 2007;doi:10.1016/j.buildenv.2007.10.010.

[35] Wang L, Chen Q. Theoretical and numerical studies of coupling multizone and CFD models for building air distribution simulations. *Indoor Air* 2007;17(5):348-361.

[36] Wang L, 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.

[37] Hu B, Freihaut JD, Bahnfleth WP, Aumpansub P, Thran B. Modeling particle dispersion under human activity disturbance in a multizone indoor environment. *Journal of Architectural Engineering* 2007;13(4):187-193.

[38] Firrantello J, Bahnfleth WP, Musser A, Freihaut JD, Jeong J-W. Use of factorial sensitivity analysis in multizone airflow model tuning. *ASHRAE Transactions* 2007;113(PART 1):642-651.

[39] Wang L, Chen Q. Applications of a coupled multizone and CFD model to calculate airflow and contaminant dispersion in built environment for emergency management. Submitted to *HVAC&R Research*.

[40] Howard-Reed C, Nabinger SJ, Emmerich SJ. Characterizing gaseous air cleaner performance in the field. *Building and Environment* 2008;43(3):368-377.

[41] Khoukhi M, Yoshino H, Liu J. The effect of the wind speed velocity on the stack pressure in medium-rise buildings in cold region of China. *Building and Environment* 2007;42(3):1081-1088.

[42] Maatouk K. A simplified procedure to investigate airflow patterns inside tall buildings using COMIS. *Architectural Science Review* 2007;50(4):365-369.

[43] Sohn MD, Apte MG, Sextro RG, Lai ACK. Predicting size-resolved particle behavior in multizone buildings. *Atmospheric Environment* 2007;41(7):1473-1482.

[44] Crawley DB, Hand JW, Kummert M, Griffith BT. Contrasting the capabilities of building energy performance simulation programs. *Building and Environment* 2008;43(4):661-673.

[45] Megri AC. Building load and energy simulation programs and the design process. *International Journal of Ventilation* 2007;6(2):177-192.

[46] Megri AC,Haghighat F. Zonal modeling for simulating indoor environment of buildings: review, recent developments, and applications. *HVAC&R Research* 2007;13(6):887-905.

[47] Song F, Zhao B, Yang X, Jiang Y, Gopal V, Dobbs G, Sahm M. A new approach on zonal modeling of indoor environment with mechanical ventilation. *Building and Environment* 2008;43(3):278-286.

[48] Yan D, Song F, Yang X, Jiang Y, Zhao B, Zhang X, Liu X, Wang X, Xu F, Wu P, Gopal V, Dobbs G, Sahm M. An integrated modeling tool for simultaneous analysis of thermal performance and indoor air quality in buildings. *Building and Environment* 2008;43(3):287-293.

[49] Jiru TE, Haghighat F. Modeling ventilated double skin façade - a zonal approach. *Energy and Buildings* 2008;doi:10.1016/j.enbuild.2008.02.017.

[50] Lukasse LJS, de Kramer-Cuppen JE, van der Voort AJ. A physical model to predict climate dynamics in ventilated bulk-storage of agricultural produce. *International Journal of Refrigeration* 2007;30(1):195-204.

[51] Norton T, Sun D-W. Computational fluid dynamics (CFD) – an effective and efficient design and analysis tool for the food industry: A review. *Trends in Food Science & Technology* 2006;17:600-620.

[52] Norton T, Sun D-W, Grant J, Fallon R, Dodd V, Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review. *Bioresource Technology* 2007;98(12):2386-2414.

[53] Launder BE, Spalding DB. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Energy* 1974;3: 269-289.

[54] Yakhot V, Orszag, SA. Renormalization group analysis of turbulence. *Journal of Scientific Computing* 1986;1(1):3-51.

[55] Smagorinsky J. General circulation experiments with the primitive equations I: the basic experiment. *Monthly Weather Review* 1963;91(3): 99-164.

[56] 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.

[57] Zhai Z, Zhang Z, Zhang W, Chen Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1 - Summary of prevalent turbulence models. *HVAC&R Research* 2007;13(6):853-870.

[58] Yuan CS. The effect of building shape modification on wind pressure differences for cross-ventilation of a low-rise building. *International Journal of Ventilation* 2007;6(2):167-176.

[59] Kuznik F., Rusaouën G, Brau J. Experimental and numerical study of a full scale ventilated enclosure: Comparison of four two equations closure turbulence models. *Building and Environment* 2007;42(3):1043-1053.

[60] Tapsoba M, Moureh J, Flick D. Airflow patterns inside slotted obstacles in a ventilated enclosure. *Computers and Fluids* 2007;36(5):935-948.

[61] Rohdin P, Moshfegh B. Numerical predictions of indoor climate in large industrial premises. A comparison between different k-ε models supported by field measurements. *Building and Environment* 2007;42(11):3872-3882.

[62] Zhao L, Wang X, Zhang Y, Riskowski GL. Analysis of airflow in a full-scale room with non-isothermal jet ventilation using PTV techniques. *ASHRAE Transactions* 2007;113(Part 1):414-425.

[63] Tian ZF, Tu JY, Yeoh GH, Yuen RKK. Numerical studies of indoor airflow and particle dispersion by large eddy simulation. *Building and Environment* 2007;42(10):3483-3492.

[64] Chang T-J, Kao H-M, Hsieh Y-F. Numerical study of the effect of ventilation pattern on coarse, fine, and very fine particulate matter removal in partitioned indoor environment. *Journal of the Air and Waste Management Association* 2007;57(2):179-189.

[65] Lai MKK, Chan ATY. Large-eddy simulations on indoor/outdoor air quality relationship in an isolated urban building. *Journal of Engineering Mechanics* 2007;133(8):887-898.

[66] Abdalla IE, Cook MJ, Rees SJ, Yang, Z. Large-eddy simulation of buoyancy-driven natural ventilation in an enclosure with a point heat source. *International Journal of Computational Fluid Dynamics* 2007;21(5-6):231-245.

[67] Zitzmann T, Cook MJ, Pfrommer P. Dynamic CFD modelling of thermal mass and air movement. *International Journal of Ventilation* 2007;6(2):145-156.

[68] Ng KC, Ng EYK, Yusoff MZ, Lim TK. Applications of high-resolution schemes based on normalized variable formulation for 3D indoor airflow simulations. *International Journal for Numerical Methods in Engineering* 2008;73(7):948-981.

[69] Shakeri A, Dolatabadi A, Haghighat F, Karimipanah T. Impact of occupant modelling on the prediction of airflow around occupants in a ventilated room. *International Journal of Ventilation* 2007;6(2):129-144.

[70] Chen Q, Zhai Z, Wang L. Computer modeling of multiscale fluid flow and heat and mass transfer in engineered spaces. *Chemical Engineering Science* 2007;62:3580-3588.

[71] Wang L, Wong NH. Coupled simulations for naturally ventilated residential buildings. *Automation in Construction* 2008;17(4):386-398.

[72] Wang L, Wong NH. The impacts of ventilation strategies and facade on indoor thermal environment for naturally ventilated residential buildings in Singapore. *Building and Environment* 2007;42(12):4006-4015.

[73] Fortmeyer R. Getting aggressive about passive design. *ENR (Engineering News-Record)* 2007;258(19):81-85.

[74] Pappas A, Zhai Z. Numerical investigation on thermal performance and correlations of double skin façade with buoyancy-driven airflow. *Energy and Buildings* 2008;40(4):466-475.

[75] Kim T, Song D, Kato S, Murakami S. Two-step optimal design method using genetic algorithms and CFD-coupled simulation for indoor thermal environments. *Applied Thermal Engineering* 2007;27(1):3-11.

[76] Gao NP, Zhang H., Niu JL. Investigating indoor air quality and thermal comfort using a numerical thermal manikin. *Indoor and Built Environment* 2007;16(1):7-17.

[77] Zhang R, Tu G, Ling J. Study on biological contaminant control strategies under different ventilation models in hospital operating room. *Building and Environment* 2008;43(5):793-803.

[78] Zhao B, Yang C, Yang X, Liu S. Particle dispersion and deposition in ventilated rooms: Testing and evaluation of different Eulerian and Lagrangian models. *Building and Environment* 2008;43(4):388-397.

[79] Zhao B, Guan P. Modeling particle dispersion in personalized ventilated room. *Building and Environment* 2007;42(3):1099-1109.

[80] Helmis CG, Tzoutzas J, Flocas HA, Halios CH, Stathopoulou OI, Assimakopoulos VD, Panis V, Apostolatou M, Sgouros G, E. Adam E. Indoor air quality in a dentistry clinic. *Science of the Total Environment* 2007;377(2-3):349-365.

[81] Sun W, Ji J, Li Y, Xie X. Dispersion and settling characteristics of evaporating droplets in ventilated room. *Building and Environment* 2007;42(2):1011-1017.

[82] Wan MP, Chao CYH. Transport characteristics of expiratory droplets and droplet nuclei in indoor environments with different ventilation airflow patterns. *Journal of Biomechanical Engineering* 2007;129(3):341-353.

[83] Zhang Z, Chen Q. Comparison of the Eulerian and Lagrangian methods for predicting particle transport in enclosed spaces. *Atmospheric Environment* 2007;41(25):5236-5248.

[84] Lai ACK, Chen FZ. Comparison of a new Eulerian model with a modified Lagrangian approach for particle distribution and deposition indoors. *Atmospheric Environment* 2007;41(25):5249-5256.

[85] Kassomenos P, Karayannis A, Panagopoulos I, Karakitsios S, Petrakis M. Modelling the dispersion of a toxic substance at a workplace. *Environmental Modelling and Software* 2008;23(1):82-89.

[86] Li J, Yavuz I, Celik I, Guffey S. Predicting worker exposure--the effect of ventilation velocity, free-stream turbulence and thermal condition. *Journal of Occupational and Environmental Hygiene* 2007;4(11):864-874.

[87] Kato S, Yang J-H. Study on inhaled air quality in a personal air-conditioning environment using new scales of ventilation efficiency. *Building and Environment* 2008;43(4):494-507.

[88] Bady M, Kato S, Huang H. Towards the application of indoor ventilation efficiency indices to evaluate the air quality of urban areas. *Building and Environment* 2008; doi:10.1016/j.buildenv.2007.11.013.

[89] Chen Q, Glicksman L, Lin J, Scott A. Sustainable urban housing in China. *Journal of Harbin Institute of Technology (New Series),* 2007;14s:6-9.

[90] Asfour OS, Gadi MB. A comparison between CFD and Network models for predicting wind-driven ventilation in buildings. *Building and Environment* 2007;42(12):4079-4085.

[91] Ji Y., Cook MJ, Hanby V. CFD modelling of natural displacement ventilation in an enclosure connected to an atrium. *Building and Environment* 2007;42(3):1158-1172.

[92] Mak CM, Niu JL, Lee CT, Chan KF. A numerical simulation of wing walls using computational fluid dynamics. *Energy and Buildings* 2007;39(9):995-1002.

[93] Li L, Mak CM. The assessment of the performance of a windcatcher system using computational fluid dynamics. *Building and Environment* 2007;42(3):1135-1141.

[94] Cheng CCK, Lam KM, Yuen RKK, Lo SM, Liang J. A study of natural ventilation in a refuge floor. *Building and Environment* 2007;42(9):3322-3332.

[95] Gosselin JR, Chen Q. A computational method for calculating heat transfer and airflow through a dual-airflow window. *Energy and Buildings* 2008;40(4):452-458.

[96] Gao J, Gao F-S, Zhao J-N, Liu J. Calculation of natural ventilation in large enclosures. *Indoor and Built Environment* 2007;16(4):292-301.

[97] Lau J, Chen Q. Floor-supply displacement ventilation for workshops. *Building and Environment* 2007;42(4):1718-1730.

[98] Zhang T, Chen Q. 2007. Novel air distribution systems for commercial aircraft cabins. *Building and Environment* 2007;42(4):1675-1684.

[99] Ng KC, Kadirgama K, Ng EYK. Response surface models for CFD predictions of air diffusion performance index in a displacement ventilated office. *Energy and Buildings* 2008;40(5):774-781.

[100] Karimipanah T., Awbi HB, Sandberg M, Blomqvist, C. Investigation of air quality, comfort parameters and effectiveness for two floor-level air supply systems in classrooms. *Building and Environment* 2007;42(2):647-655.

[101] Yuan F-D, You S-J. CFD simulation and optimization of the ventilation for subway side-platform. *Tunnelling and Underground Space Technology* 2007;22(4):474-482.

[102] Bin Y, Sekhar SC. Three-dimensional numerical simulation of a hybrid fresh air and recirculated air diffuser for decoupled ventilation strategy. *Building and Environment* 2007;42(5):1975-1982.

[103] Khan MI. Effects of nozzle geometry on air flow jet and temperature distribution in an enclosed space. *International Journal of Ventilation* 2007;5(4):405-415.

[104] Zerihun Desta T, Baelmans M, Berckmans D. Behaviour of well-mixed zones (WMZs) in an imperfectly mixed ventilated room. *Building and Environment* 2008; doi:10.1016/j.buildenv.2007.12.002.

[105] Zhang T, Chen Q, Lin C-H. Optimal sensor placement for airborne contaminant detection in an aircraft cabin. *HVAC&R Research* 2007;13(5):683-696.

[106] Mazumdar S, Chen Q. Influence of cabin conditions on placement and response of contaminant detection sensors in a commercial aircraft. *Journal of Environmental Monitoring* 2008;10:71-81.

[107] Lomas KJ.Architectural design of an advanced naturally ventilated building form. *Energy and Buildings* 2007;39:166–181.

[108] Marsik T, Johnson R. Use of a state space model to study the effect of outdoor air quality on indoor air in Fairbanks Alaska. *Indoor and Built Environment* 2007;16(6)538–547.

[109] Pietrzyk K, Hagentoft C-E. Probabilistic analysis of air infiltration in low-rise buildings. *Building and Environment* 2008(4);43:537–549.

[110] Zhang T, Chen Q. Identification of contaminant sources in enclosed environments by inverse CFD modelling. *Indoor Air* 2007;17(3):167-177.

[111] Zhang T, Chen Q. Identification of contaminant sources in enclosed spaces by a single sensor. *Indoor Air* 2007;17(6):439-449.

[112] Liu X, Zhai Z. Location identification for indoor instantaneous point contaminant source by probability-based inverse computational fluid dynamics modeling. *Indoor Air* 2008;18(1);2-11.