Wang, L. and Chen, Q. 2007. "Validation of a coupled multizone and CFD program for building airflow and contaminant transport simulations," *HVAC&R Research*, 13(2), 267-281.

Validation of a Coupled Multizone and CFD Program for Building Airflow and Contaminant Transport Simulations

Liangzhu Wang
Student Member ASHRAE

Qingyan Chen, PhD[1]
Fellow ASHRAE

ABSTRACT

Current multizone airflow network models assume air momentum effects, contaminant concentrations and air temperatures are uniformly and homogeneously distributed in a zone of a building. These assumptions can cause errors for zones where air and/or contaminant are not well mixed. A coupled multizone-CFD program has been developed to improve the multizone model by applying a CFD model to those poorly-mixed zones and the multizone model to the rest zones. This paper validated the coupled multizone-CFD program by using experimental data obtained in a four-zone facility with non-uniform distributions of air momentum effects, contaminant concentrations, and air temperatures. The calculated results by the coupled program generally agreed with the experimental data although discrepancies exist in some cases. The coupled multizone and CFD simulations used less computing time than the CFD simulations for the whole flow domain.

Key words: Multizone, CFD, Coupling, Simulations, Experimental validation

INTRODUCTION

Multizone airflow network models and Computational Fluid Dynamics (CFD) have been widely used in simulations of building airflow distribution and contaminant transport (Spengler and Chen 2000; Emmerich 2001). Multizone models focus on the average airflow characteristics and contaminant dispersion caused by infiltration. The computing costs are minimal since it only takes several minutes to perform an hour-by-hour dynamic simulation of a whole building for one year. Nevertheless, to achieve fast computational speed, multizone models have to use many assumptions. For example, multizone network models assume that each room of a building is a zone with a uniform temperature and contaminant concentration. They also neglect airflow momentum preserved inside a zone. These assumptions may compromise the accuracy of the results of the simulations. Compared to multizone models, CFD methods use fewer assumptions and are able to obtain detailed spatial values of airflow and contaminant concentrations. One of the drawbacks of CFD is that it demands a lot more computing time than the multizone models.

[1] **Liangzhu Wang** is a PhD candidate and research assistant and **Qingyan Chen** is a professor in the School of Mechanical Engineering, Purdue University, West Lafayette, IN.

A simulation of a whole building at steady-state conditions can take several hours or even days. Therefore, CFD is not feasible for hour-by-hour dynamic simulations for a year.

It seems that the coupling of a multizone method with a CFD method can combine their merits and avoid their drawbacks. One could use CFD to those zones where the uniform assumptions would fail so the accuracy of the simulated results can be greatly improved, compared with using a multizone model alone. On the other hand, CFD is applied to limited zones so that the computing time with the coupled program would be more manageable, compared with using a CFD model alone for the whole building.

Some previous studies have shown promising results of coupled multizone-CFD simulations. Schaelin et al. (1994) proposed a "method of detailed flow path values," in which the perfect mixing assumptions of multizone models were remedied by providing detailed contaminant concentrations at the flow paths from the CFD results. Their test cases showed that the calculated concentrations from this technique differ by a factor of up to 2.5 from the pure multizone approach. Clarke et al. (1995) and Negrao (1998) implemented an internal coupling for airflow simulations in ESP-r (Clarke 1985). The coupled program could provide non-uniform distributions of airflow and contaminant concentrations, which otherwise could not be obtained from pure multizone simulations. Jayaraman et al. (2004) also showed that the calculated personal exposure to contaminant by a coupled multizone-CFD approach could be quite different from a multizone method, especially for buildings with large spaces. However, these studies did not validate their results with experimental data, without which it is hard to determine whether the coupled multizone-CFD approach could provide more realistic results than the pure multizone method. Yuan (2003) conducted the first on-site experimental validation of the coupled multizone-CFD approach, which showed good results compared to the measured data. It is rather difficult to obtain such data on site and the data could be used for the present study. Since the authors were not aware of the data in the thesis until we revised this paper, this study conducted independent experimental measurements of airflow distribution for a four-zone building model that used controlled boundary conditions.

Thus, it is necessary to perform systematic validation of a coupled multizone and CFD program to access its performance in simulating cases where the multizone assumptions fail. It is also important to validate the coupled program under more realistic flow conditions rather than predefined flow rates. These form the main objectives of the research results published in this paper.

COUPLING APPROACH OF A MULTIZONE MODEL WITH A CFD MODEL

A multizone network model, such as CONTAM (Walton and Dols 2003), calculates the 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 from zone i to zone j, F_{ij} is often calculated by a multizone model by a power-law function of the pressure drop, ΔP_{ij} , across path ij,

$$F_{ij} = \alpha_{ij} c_{ij} \,|\, \Delta P_{ij} \,|^{n_{ij}} \tag{1}$$

where, c_{ij} is the flow coefficient, n_{ij} is the flow exponent of Path ij, and α_{ij} is "+1" for the airflow from zone i to zone j and "-1" for the airflow in the opposite direction. For each zone, multizone models solve air mass balance equations under steady state condition for zone j as

$$\sum_i F_{ij} + F_j = \sum_i \alpha_{ij} c_{ij} \,|\, \Delta P_{ij} \,|^{n_{ij}} + F_j = 0 \tag{2}$$

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

$$\sum_i F_{ij} C_i - \sum_i F_{ji} C_j + S_j = 0 \tag{3}$$

where, C_i and C_j are the contaminant concentrations in zone i and zone j, respectively; F_{ji} is the airflow rate from zone j to zone i; and S_j=contaminant sources inside zone j.

To close the equation system, multizone models use the following assumptions:

- Uniform pressure at the same height of a zone, uniform temperature and uniform contaminant concentration in a zone
- Quiescent or still air in a zone; airflow through zones does not impact zone pressure
- Momentum and kinetic energy not accounted for by flow path models

These assumptions may compromise the accuracy of the results obtained with a multizone model. Upham (1997) pointed out that, for non-uniform distributions of contaminant concentrations, the results of using a multizone model were questionable. Clarke (2001) also noted that current buildings' airflow modeling-by-network approach has significant limitations of determining intra-room airflow and temperature distribution correctly. Gao and Chen (2003) found that multizone models produce incorrect results due to the neglected momentum within a zone. To improve the simulation results, more sophisticated models, such as CFD methods, should be used.

The Reynolds Averaged Navier-Stokes (RANS) modeling with turbulence models is one of the most popular CFD methods for calculating airflow and contaminant transport in buildings as reviewed by Emmerich (1997), Ladeinde and Nearon (1997), and Nielsen (1998). A RANS model, such as CFD0 (Srebric, Chen et al. 1999), solves a set of partial differential governing equations for mass, momentum, energy, and species conservation. The equation can be written in a general form (Patankar 1980):

$$\sum a_{\phi,nb}\phi_{nb} - a_{\phi,P}\phi_P + b_\phi = 0 \quad (4)$$

When ϕ stands for pressure (P), it is the mass continuity equation; for air velocity component (U_i, U_j, and U_k), it is the momentum equations; for temperature (T), it is the energy conservation equation; and for species concentration (C), it is the concentration conservation equation.

This study has coupled CONTAM and CFD0 programs by applying CFD0 to the zones, where the multizone assumptions fail, and CONTAM to the rest zones. The coupled CONTAM-CFD0 program solves an assembled matrix equation of airflow simulations. By linearizing and combining both Eqs. (2) and (4) for P, we obtain the following assembled airflow equation:

$$\mathbf{CP} + \mathbf{F} = \mathbf{B} \quad (5)$$

where, **C** is the airflow coefficient matrix; **P** is the pressure vector of zones and cells; **F** is the flow rate vector of paths and cells at multizone and CFD zone interfaces; **B** is the vector of air mass sources.

The coupled program solves Eq. (5) iteratively in which CONTAM gives pressure boundary conditions to CFD0 and CFD0 returns pressure boundary conditions to CONTAM. The solving procedure is realized through a mode of “CONTAM – CFD0 – CONTAM” so that the output of one program becomes the input of the other. When both inputs and outputs stabilize (their values do not change), the solution of the coupling is considered convergent. (Wang and Chen 2005).

EXPERIMENTAL DESIGN

To validate the coupled CONTAM-CFD0 program, the experiment used an environmental chamber facility at Purdue University. The coupled program will apply CFD0 to zones, where CONTAM assumptions fail, and apply CONTAM to the rest zones so the chamber facility was partitioned further into four zones as shown in Fig. 1. This paper validated the coupled program for both airflow and contaminant transport simulations. For airflow simulations, this investigation measured the wall temperatures, spatial distributions of air velocities, air temperatures, and airflow rates through the openings between the zones. Eighteen thermal couples were embedded in the walls for temperature measurements, supplemented with an infrared thermometer. The spatial air velocities and temperatures were measured at 63 locations with omni-directional hot-sphere anemometers. Contaminant concentrations were simulated by sulfur hexafluoride gas (SF_6), which was measured at 45 locations by a tracer gas analyzer.

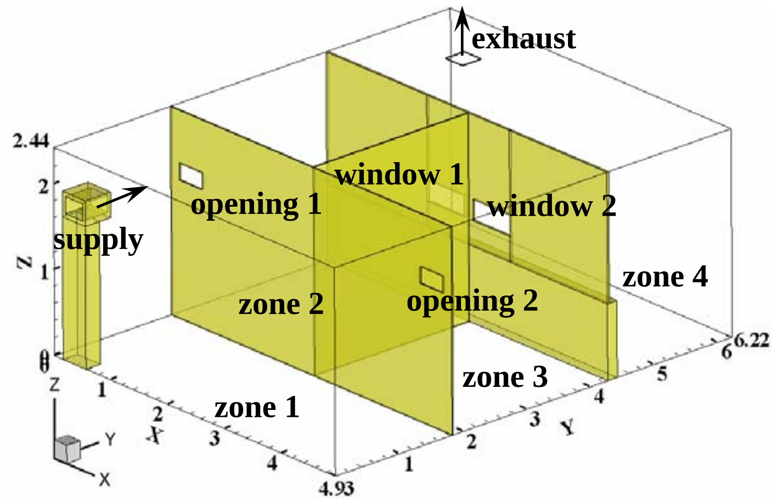


Fig. 1. Schematic of the chamber facility divided into four zones used in the study.

The airflow rates through internal openings and windows were measured indirectly through the steady-state tracer gas method (Etheridge and Sandberg 1996). The equilibrium flow rate through an opening was calculated by:

$$Q = \frac{\dot{m}}{\bar{C}} \qquad (6)$$

where, Q is the flow rate through an opening, $\dot{m}$ is the tracer-gas source (SF_6) flow rate, and $\bar{C} = (\sum_i C_i A_i)/A$, that is the average SF_6 concentration through the opening.

VALIDATION OF CONTAM-CFD0 PROGRAM

In order to validate the coupled CONTAM-CFD0 program and to show that the coupled program is superior to a multizone method, the experiment should study the situations where the CONTAM assumptions fail. Based on the previous discussions of multizone assumptions, this paper studied airflows with

- non-uniform air momentum distributions;
- non-uniform contaminant concentration distributions; and/or
- non-uniform air temperature distributions.

This section would show how this investigation used the four-zone facility to create the three scenarios. The experimental data measured from the facility is then used to demonstrate whether a multizone model, such as CONTAM, would give accurate simulations for the three scenarios. If the answer is negative, then whether a coupled multizone and CFD model, such as CONTAM-CFD0, could improve the accuracy of the simulations.

Non-uniform Air Momentum Distributions

CONTAM assumes that the air in a zone is quiescent or still. The assumption is valid for zones with very low air velocity, such as airflow caused by infiltration through cracks. In such a case, the infiltration is immediately dissipated after entering the zone. However, a strong momentum effect may be preserved, contributing to spatial variations in zone pressures, if the airflow is from a large opening with a high air velocity. Then the inflow momentum effect could not be dissipated completely in the zone.

In order to create a strong momentum effect in a zone in the experimental facility, this investigation used mechanical ventilation through an air supply in Zone 1, as shown in Fig. 1. Zone 1 had two openings connecting to the neighboring zones. One connected with Zone 2 was on the opposite side of the supply and the other to Zone 3 on the other end that was far from the supply.

The supply opening size was 0.3 m × 0.2 m, with an effective area ratio of 0.77. The airflow rates used in the experiment were 73, 113, 223, 296, and 456 CFM (or 0.034, 0.053, 0.105, 0.14, and 0.215 m^3/s). Since it is hard for CFD0 to simulate recirculation flows across a large opening, the opening size was carefully selected to make sure that only one-way flow was caused. The size of Openings 1 and 2 was 0.40 m × 0.20 m each and the other openings in Zones 2 and 3 (Windows 1 and 2) were 0.65 m × 0.20 m. The design of one-way flows also made it possible to use the orifice airflow model for Openings 1 and 2

$$F = C_D A \sqrt{2\Delta P / \rho} \qquad (7)$$

where, C_D is the discharge coefficient for opening, A is the cross-sectional area of opening, ΔP is the pressure difference across opening, and ρ is the air density. After converting Eq. (7) to Eq. (1), we obtained that the flow coefficient, c_{ij}, and the flow exponent, n_{ij}, was 0.076 and 0.5, respectively for Openings 1 and 2.

As shown in Fig. 2, the airflows measured through Openings 1 and 2 were not equal, although the geometry is symmetrical. This is because the momentum effect from the air supply brought more air to Opening 1 than that to Opening 2. The higher the momentum (airflow rate) from the supply, the higher the ratio of the flow through Opening 1 over Opening 2.

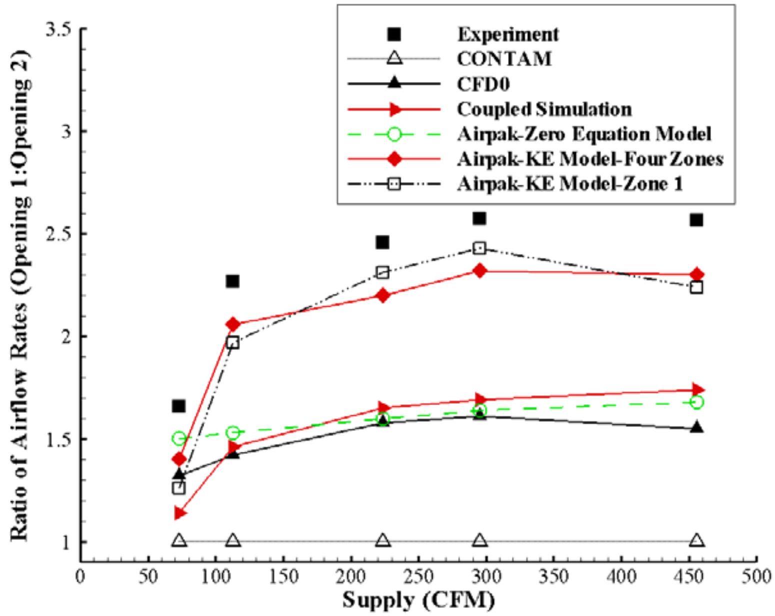


Fig. 2. Comparison of the calculated airflow ratio through Opening 1 over Opening 2 with different methods with the measured data for the non-uniform momentum case (1 CFM = 0.0005 m^3/s).

A multizone model, such as CONTAM, could not consider the non-uniform momentum effect in Zone 1 created by the supply air. As a result, the airflow rate through the two openings calculated by CONTAM would be the same, as shown in Fig. 2. In reality, Opening 1 would have a higher flow rate than Opening 2 caused by the momentum effect from the air supply, as the zone geometry in the downstream was almost symmetrical. Thus, the multizone model fails to calculate accurately airflow distribution for this case with non-uniform momentum flow.

When the coupled CONTAM-CFD0 program was used, where CFD0 was applied to Zone 1 and CONTAM for the rest zones, Fig. 3 shows that the pressure near Opening 1 was much greater than that near Opening 2. As a result of the non-uniform distribution of pressure in Zone 1, the flow rate through Opening 1 was greater than that through Opening 2. However, the discrepancies between the calculated results and the measured data were very significant as shown in Fig. 2. These results were consistent with different airflow rate and were unexpected.

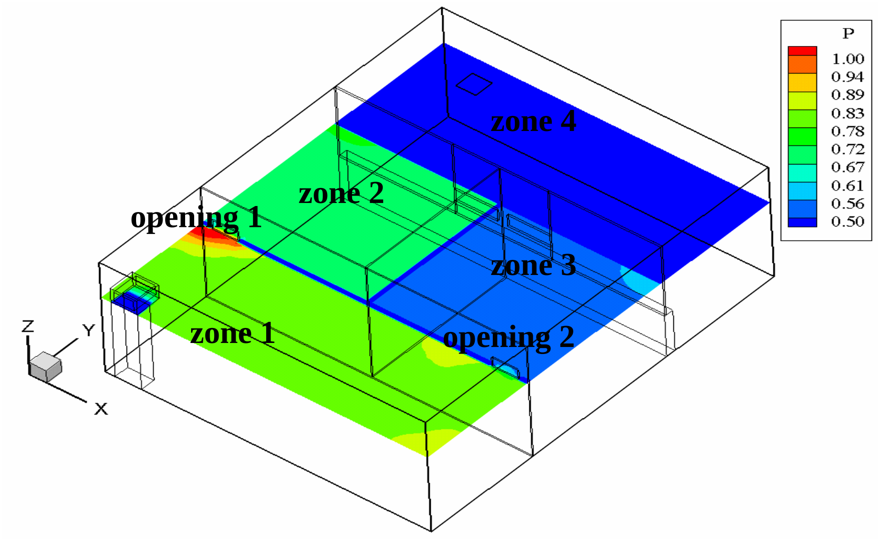


Fig. 3. The pressure distribution at 1.66 m from the floor in the four-zone chamber calculated by CFD0.

By applying CFD0 to all the four zones, the calculated airflow ratios did not improve further, compared with those obtained with the coupled CONTAM-CFD0 program. Then, a commercial CFD program, Airpak (Fluent 2002), was applied to all the four zones by using the same zero-equation turbulence model as what used in CFD0. The results obtained by Airpak were almost identical to those by CFD0. Clearly, for this particular case, the zero-equation model was unable to predict accurately the airflow in the four-zone facility, although the reason was not clear.

If we replaced the zero-equation model by the standard k-ε model in Airpak and simulated all the four zones, the computed ratios of airflow rates of Opening 1 over Opening 2 were very close to the experimental data as also shown in Fig. 2. This further verified that the turbulence model played a very important role in this case. Finally, this investigation applied Airpak with the standard k-ε model to Zone 1 and CONTAM to the other three zones. The computed results were in reasonable agreement with the experimental data and those obtained by using Airpak with the standard k-ε model to all the zones. This validates that the coupled multizone and CFD model can be used for the prediction of airflow in zones with non-uniform momentum distributions.

For this case, the computation time is about 490 minutes for the simulation with CFD0 for all the four zones with a total grid of 70×73×33 (X×Y×Z) and 185 minutes for the coupled CONTAM-CFD0 simulation with a total grid of 70×27×33 (X×Y×Z) for Zone 1. The coupled simulation used less computing time than the CFD0 simulation since it applied CFD0 to Zone 1 only. Although the computing cost was much greater than CONTAM, which only takes a few seconds, the coupled simulation provided more accurate results than CONTAM. The simulations with the standard k-ε turbulence model required more computing time than those with the zero-equation model. An Airpak simulation with the same grid number as CFD0 for the four zones by the standard k-ε model cost 620 minutes, which doubled the simulation time with the zero-equation model in Airpak.

In addition to flow rates, our experiment also measured the air velocity and temperature in 63 locations in Zones 1, 2, and 3 of the four-zone facility. The 63 locations were in nine poles as illustrated by P1 through P9 in Fig. 4, and each pole has seven points from the floor to the ceiling.

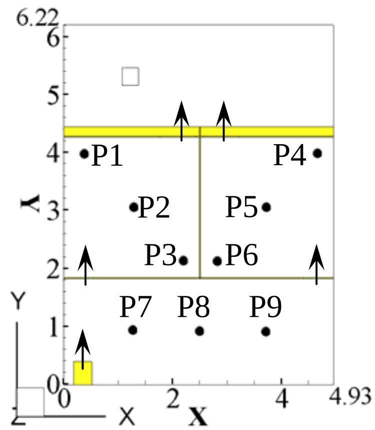
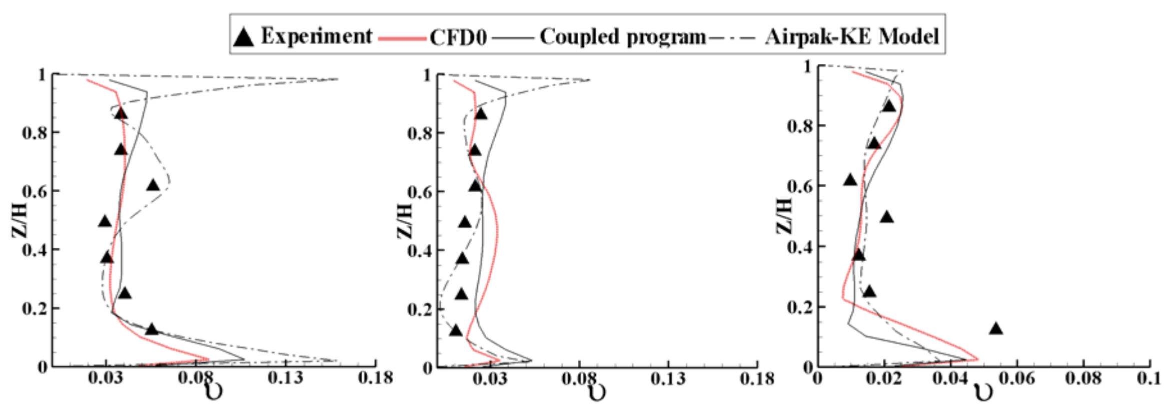


Fig. 4 The plan view of the four-zone facility with the nine measuring poles

Fig. 5 compares the air velocities calculated by CFD0, the coupled CONTAM-CFD0, and the Airpak simulation of the four zones by the standard k-ε model for selected positions with the experimental data. Since the coupled CONTAM-CFD0 simulation applied CFD0 only to Zone 1, the comparison can only be made for P7, P8, and P9. The Airpak simulation with the standard k-ε model provided the best results while the CFD0 results had some discrepancies from the experimental data. The discrepancies could be attributed to the turbulence model, as discussed previously. The calculated air velocities by the coupled program generally agree well with those by the CFD0, although similar degree of discrepancies was found. Even if the coupled simulation only applied CFD0 to one zone, the coupled CONTAM-CFD0 performed as well as CFD0 for all the zones.



(a) P7 (b) P8 (c) P9

Fig. 5 Comparison of air velocities at P7, P8, and P9 in Zone 1 with non-uniform momentum distribution (H=2.44m; υ =U/U_{in}, non-dimensional velocity when U_{in}=2.281 m/s is the supply air velocity and the supply rate is 223 CFM (0.112 m^3/s)

Non-Uniform Contaminant Concentration Distributions

CONTAM assumes instantaneous mixing of a contaminant in a zone. Such an assumption is acceptable if the zone is small and the mixing is intensive. In many cases, the mixing is not perfect. By applying CFD to such a zone, the non-uniform mixing can be considered so that the simulated results could be greatly improved.

Fig. 6 shows the chamber schematic to study non-uniform contaminant mixing. The case was the same as that for the study of non-uniform momentum distributions, except for adding a partition wall in front of the air supply in Zone 1. This study created a non-uniform SF_6 distribution in Zone 1 by placing a contaminant source, which was simulated by SF_6, behind the partition.

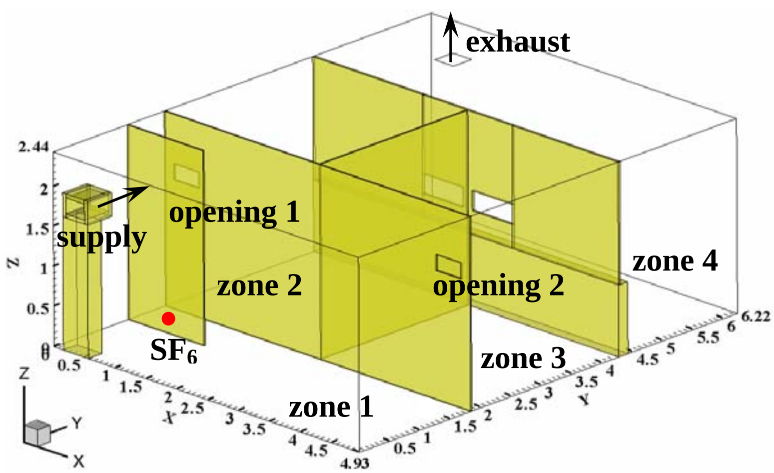


Fig. 6. The chamber schematic used for the non-uniform contaminant distribution case.

It was possible to improve CONTAM results by separating the area behind the partition wall as another zone. However, the airflow between the new zone and the rest of Zone 1 could be multi-directional through the large opening. That creates a very challenging problem for CFD0. On the other hand, our aim is to demonstrate how CFD0 could be used to improve the airflow modeling not to demonstrate how we could do better by using CONTAM alone.

Thus, if a CONTAM simulation is applied for the whole space of zone 1, it could not predict the non-uniform distribution of SF_6 in Zone 1. CONTAM also interprets the flow and geometrical conditions to be symmetrical, because it could not take the partition wall into account. As a result, CONTAM would predict the same SF_6 concentration in Zone 2 and Zone 3, as shown in Fig. 7.

Without concerning about airflow models, the coupled CONTAM and CFD0 simulation, in which the CFD0 was applied to Zone 1, could consider the non-uniform SF_6 concentration in Zone 1. Clearly, the zone next to the SF_6 source had a high SF_6 concentration than the other zone, although the geometry is symmetrical. The SF_6 concentration in Zone 2 was 0.977 ppm, which was obtained by averaging the SF_6 concentration over the 15 locations along Poles 1-3. The average SF_6 concentrations were 0.022 ppm for Zone 1 and 0.018 ppm for Zone 3. Most of the SF_6 was transported to Zone 2 so the average concentration in Zone 1 was low.

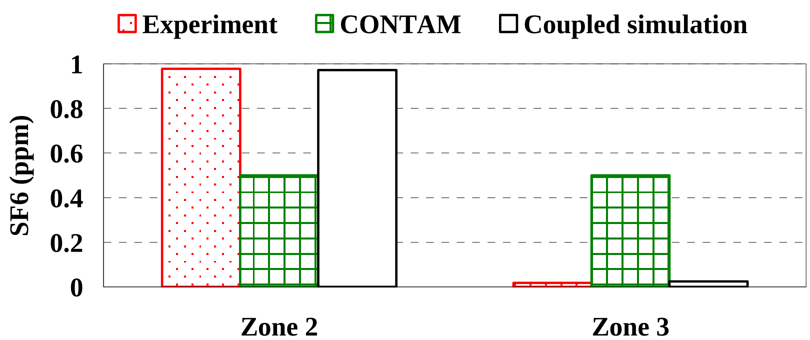


Fig. 7. Comparison of measured and simulated SF_6 concentrations (ppm) in Zones 2 and 3 for the non-uniform contaminant distribution case.

The huge difference in SF_6 concentrations between Zones 2 and 3 was caused by the non-uniform SF_6 distribution in Zone 1, as illustrated in Fig. 8. The partition wall confined the SF_6 in a corner. As a result, the SF_6 concentration near Opening 1 was about 50 times higher than that near Opening 2. The coupled program can correctly calculate the SF_6 concentrations by predicting the detailed SF_6 distribution in Zone 1.

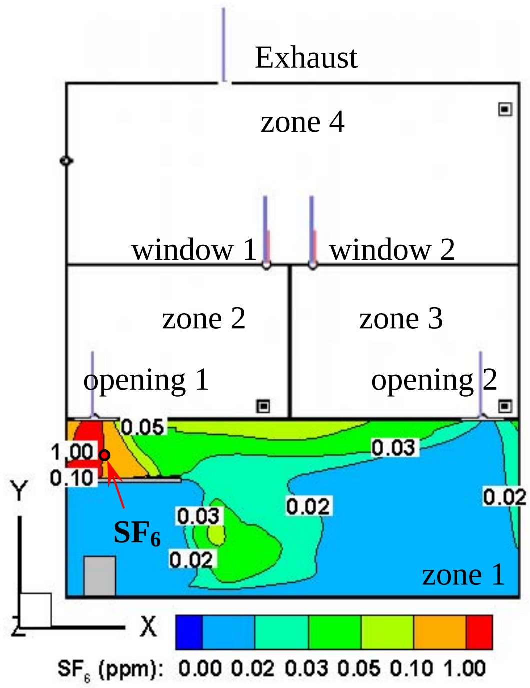


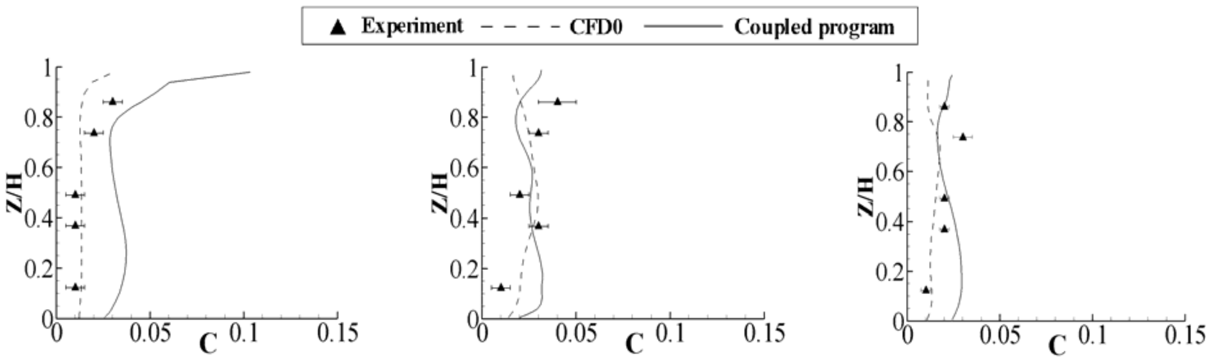
Fig. 8 SF_6 concentration distribution at Openings 1 and 2 height in the four-zone chamber facility calculated by the coupled CONTAM and CFD0 program for the non-uniform contaminant distribution case.

Fig. 8 also shows the airflow distributions from the coupled simulation. The blue lines stand for airflow rates and the red lines for pressure differences across the airflow. The partition board in front of the supply prevented the development of the inflow momentum effect in Zone 1 so that the zero-equation turbulence model performed better than the previous case. The inflow air was almost equally distributed between Openings 1 and 2. Table 1 illustrates that the calculated airflow rates through openings 1 and 2 were close to the measured data.

Table 1 Comparison of the measured airflow rate in CFM through Openings 1 and 2 with that by the coupled simulation for the non-uniform contaminant distribution case (1 CFM = 0.0005 m^3/s).

	Experiment	Coupled simulation
Opening 1	104	102
Opening 2	106	108

Fig. 9 compares the SF_6 concentrations in Zone 1 obtained with different methods. The SF_6 concentrations were measured at 45 locations at the same nine poles as in the previous case. When applying CFD0 to the entire flow domain (all the four zones), the CFD0 results were reasonably close to the experimental data, although some fluctuations of experimental data existed. Since it took 30 s to measure the SF_6 concentration in one location, the data obtained between the two measurements that were two minutes apart may not be the same for the same location. The solid triangles show the mean SF_6 concentration measured and the horizontal bars show the fluctuation. Our experience also shows that it is very difficult to obtain good agreement between CFD simulations and experimental measurements for tracer-gas concentration. The coupled CONTAM and CFD0 simulation predicted higher SF_6 concentrations for Poles 7-9. This is probably caused by the inaccurate flow rate provided by CONTAM in other zones for the CFD0 as boundary conditions.



(a) P7 (b) P8 (b) P9

Fig. 9. Comparison of measured and computed SF_6 concentrations (ppm) at P7, P8, and P9 in Zone 1 for the non-uniform contaminant distribution case.

Non-Uniform Air Temperature Distributions

A multizone model, such as CONTAM, could consider the impact of temperature on the airflow between zones. However, the model assumes that the temperature is uniformly distributed in each zone. Therefore, the impact caused by the temperature gradient in each zone is not accounted for. Hence, this investigation designed another case with a temperature gradient to examine the impact of the temperature gradient on the airflow distributions calculated by CONTAM.

Fig. 10 shows the schematic of the case designed. A second air supply was added in Zone 1 to create a symmetrical flow-supply condition. A heated box was placed in Zone 2 and a non-heated box of the same size was symmetrically placed in Zone 3. This experiment also placed Openings 1 and 2 in the lower part of the partition to enhance stack effect for Zone 2. Because Zone 2 was of a higher temperature than Zone 3 due to the heat source, the stack effect created a higher flow into Zone 2 than Zone 3.

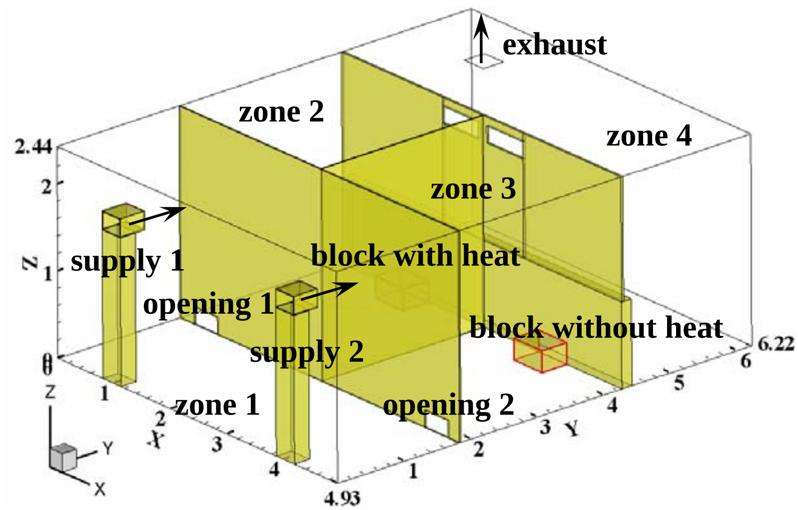


Fig. 10. Schematic of the case used to study the non-uniform temperature distribution case.

Fig. 11 shows that the measured airflow ratios through Opening 1 over Opening 2 increased with the stack effects in Zone 2, which were caused by increasing the surface temperatures of the heated box. As shown in Fig. 11, CONTAM generally under-predicted the airflow ratios by 10%, although it could consider the stack effects caused by the temperature difference of Zones 2 and 3. The reason was that CONTAM neglected the temperature gradients inside Zone 2. To consider the temperature gradients, this study applied CFD to Zone 2 and CONTAM to the rest zones in the coupled CONTAM-CFD0 simulations. The measured surface temperatures of the heated box were used as boundary conditions in the coupled simulations so that the temperature gradients could be correctly considered. Fig. 11 illustrates that the calculated airflow ratios by the coupled program were very close to the measured data except for the box surface temperature of 30°C. This study then used CFD0 to simulate all the four zones and found that the results of CFD0 were close to those of the coupled simulations for all the three cases. When the surface temperature was 30 °C, therefore, the discrepancy between the measured and calculated results could be attributed to the experimental errors.

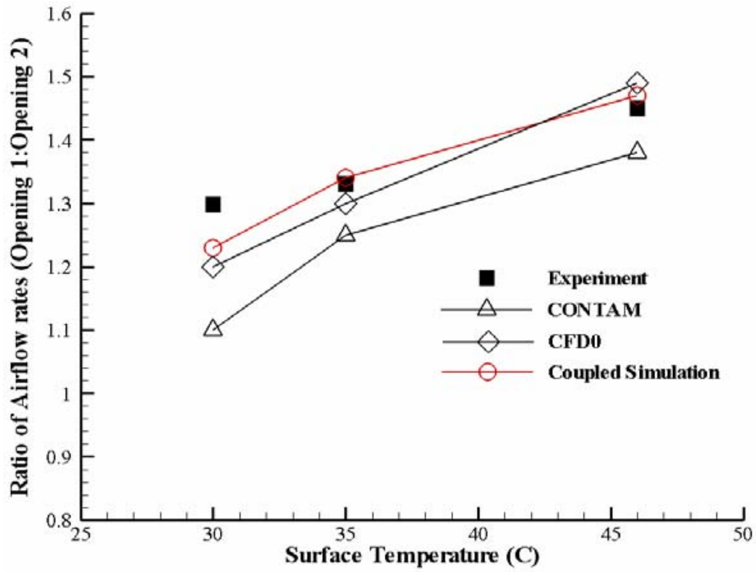
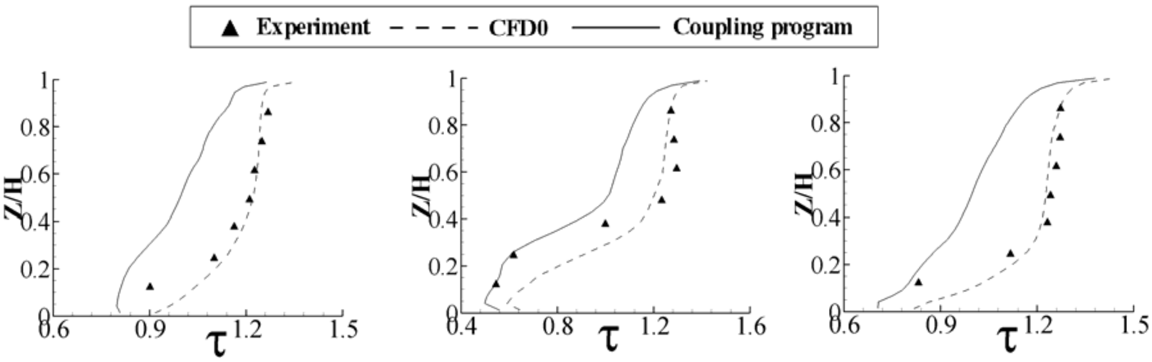


Fig. 11 Comparison of measured airflow rates with simulated results for the case of non-uniform temperature distributions with different surface temperatures of the heated box in Zone 2

On the other hand, the assumption of uniform temperature in each zone seems to be tolerable in this case. The difference of 10% of CONTAM simulations from the experimental data was within the normal acceptable range of 20% for multizone simulations (Emmerich 2001). Of course, the experiments provided CONTAM simulations a good estimation of air temperature for each zone. Otherwise, the difference of 10% may be difficult to obtain.

The moderate temperature gradients in this case explained why the assumption of uniform temperature was not crucial. Fig. 12 illustrates the temperature distribution of Poles 1-3 of Zone 2 when the surface temperature of the heated box was 35 °C. The measured temperature gradient of the bulk air in Zone 2 was as high as 3.7 °C, although the temperature gradient near the heated box could be higher. The temperature gradient could reach to 5.5 °C when the surface temperature of the heated box was 46 °C. Li et al. (1998) also found that when the temperature gradient was moderate, there was reasonable agreement between the ventilation flow rates predicted by multizone and CFD approaches. However, when the temperature gradient was more than 10 °C, the calculated ventilation rates by multizone methods can differ from the measured data by more than 30% (Kotani, Satoh et al. 2003).



(a) P1 (b) P2 (c) P3

Fig. 12 Comparison of the air temperatures at the three poles in Zone 2 for the non-uniform temperature distribution when the surface temperature of the heated box was 35 °C (H=2.44m; $\tau = (T-T_{in})/(T_{out} - T_{in})$ is non-dimensional temperature where T_{in}=18.2 °C was the supply air temperature and T_{out}=24.3 °C was the exhaust air temperature)

Fig. 12 also compares the calculated temperature distributions of Poles 1-3 with the measured data. Since the coupled simulation only applied CFD to Zone 2, the calculated temperatures were quite different from the measured data, although the pattern of the temperature differences was similar to the data. The results could be improved if CFD0 was applied to all the four zones as also shown in Fig. 12. However, the computing time of the CFD0 simulation for all four zones was one-order magnitude greater than the coupled program. When the accuracy of the spatial temperatures was not a primary concern, the coupled program provides acceptable results.

Note that, for the purpose of experimental validation of the coupled CONTAM-CFD0 program, this paper used extreme cases of non-uniform air momentum effects and contaminant concentrations. In fact, our experience shows that CONTAM would work well for most of indoor airflow simulations. Our on-going study is trying to develop guidelines when and what case such coupling is needed and the results will be reported in the near future.

CONCLUSIONS

This paper validated a coupled multizone-CFD program with experimental data obtained from a 4-zone chamber facility for the simulations of non-uniform distributions of air momentum effects, contaminant concentrations, and air temperatures. The coupled program is to improve the simulations of a multizone program, CONTAM, by coupling it with a CFD program, CFD0, when the well-mixing assumptions of CONTAM fail.

For airflows with a strong air momentum effect in a zone, the coupled program applied CFD0 to the zone with the strong momentum effect and CONTAM to the rest zones. Compared to CONTAM simulations, the coupled program calculated more accurate airflow rates. Compared with a CFD0 simulation, the coupled program used less computing time but the calculated airflow rates were lower than the measured data, which was attributed to the zero-equation turbulence model used. The results could be improved by using the standard k-ε model with yet higher computational cost than the simulations with the zero-equation model.

This paper also validated the coupled CONTAM-CFD0 program for simulating non-uniform distributions of contaminant concentrations and air temperatures in buildings. With up to one order of magnitude less of computational time than CFD0 simulation for all the four zones, the coupled simulations can predict correctly the airflow and contaminant distributions. This investigation also found that the assumption of uniform zone temperature in a multizone model is acceptable when the air temperature gradient in a zone is moderate and the air temperature at each zone can be correctly estimated.

ACKNOWLEDGEMENT

This research was supported by U.S. National Institute of Standards and Technology (NIST) through contract SB1341-04-Q-0771. The authors would like to thank Dr. A.K. Persily and Mr. G.N. Walton of NIST for their valuable contributions to the research.

REFERENCES

Clarke, J. A. 1985. *Energy Simulation in Building Design*. Bristol: Adam Hilger Ltd.

Clarke, J. A. 2001. Domain integration in building simulation. *Energy and Buildings* 33(4): 303-308.

Clarke, J. A., W. M. Dempster and C. O. R. Negrao. 1995. The implementation of a computational fluid dynamics algorithm within the ESP-r System. Building Simulation 95 Conference, Madison, Wisconsin, USA, IBPSA (The international Building Performance Simulation Association).

Emmerich, S. J. 1997. Use of computational fluid dynamics to analyze indoor air quality issue. Gaithersburg, MD, National Institute of Standards and Technology.

Emmerich, S. J. 2001. Validation of multizone IAQ modeling of residential-scale buildings: a review. *ASHRAE Transactions* 107(2): 619-628.

Etheridge, D. and M. Sandberg. 1996. *Building Ventilation: Theory and Measurement*. West Sussex, England: John Wiley & Sons.

Fluent. 2002. Airpak user manual, Fluent Inc. and ICEM-CFD Engineering.

Gao, Y. and Q. Chen. 2003. Coupling of a multi-zone airflow analysis program with a computational fluid dynamics program for indoor air quality studies. The 4th International Symposium on HVAC, Symposium on HVAC, Tsinghua University, Beijing, China.

Jayaraman, B., D. Lorenzetti and A. Gadgil. 2004. Coupled model for simulation of indoor airflow and pollutant transport. Berkeley, CA, USA, Lawrence Berkeley National Laboratory.

Kotani, H., R. Satoh and T. Yamanaka. 2003. Natural ventilation of light well in high-rise apartment building. *Energy and Buildings* 35(4): 427-434(8).

Ladeinde, F. and M. Nearon. 1997. CFD applications in the HVAC&R industry. *ASHRAE Journal* 39(1): 44-48.

Li, Y., D. A, S. JG and C. L. 1998. Comparison of zonal and CFD modeling of natural ventilation in a thermally stratified building. The 6th International Conference on Air Distribution in Rooms, Stockholm.

Negrao, C. O. R. 1998. Integration of computational fluid dynamics with building thermal and mass flow simulation. *Energy and Buildings* 27(2): 155-165.

Nielsen, P. V. 1998. The selection of turbulence models for prediction of room airflow. *ASHRAE Transactions* 104 (B): 1119-1127.

Patankar, S. V. 1980. *Numerical heat transfer and fluid flow*. New York: McGraw-Hill Book Company.

Schaelin, A., V. Dorer, J. v. d. Maas and A. Moser. 1994. Improvement of multizone model predictions by detailed flow path values from CFD calculations. *ASHRAE Transactions* 100(Part 2): 709-720.

Spengler, J. D. and Q. Chen. 2000. Indoor air quality factors in designing a healthy building. *Annual Review of Energy and the Environment* 25: 567-600.

Srebric, J., Q. Chen and L. R. Glicksman. 1999. Validation of a zero-equation turbulence model for complex indoor airflows. *ASHRAE Transactions* 105(2): 414-427.

Upham, R. 1997. A validation study of the airflow and contaminant migration computer model CONTAM as applied to tall buildings. University Park, Pennsylvania, the Pennsylvania State University.

Walton, G. N. and W. S. Dols. 2003. *CONTAMW 2.1 user manual*. Gaithersburg, MD, USA: National Institute of Standards and Technology.

Wang, L. and Q. Chen. 2005. On solution characteristics of coupling of multizone and CFD programs in building air distribution simulation. Building Simulation'05, Montreal, Canada.

Yuan, J. 2003. Effective prediction of air distribution and contaminant transport in entire buildings by coupling multizone, CFD and energy models. University Park, Pennsylvania, The Pennsylvania State University