

Srebric, J. and Chen, Q. 2002. "Simplified numerical models for complex air supply diffusers," *HVAC&R Research*, 8(3), 277-294.

Simplified Numerical Models for Complex Air Supply Diffusers

Jelena Srebric*, Ph.D.
Member ASHRAE

Qingyan Chen, Ph.D.
Member ASHRAE

ABSTRACT

Correct description of the flow and thermal information of an air supply diffuser is fatally important for a reliable prediction of room air distribution by using computational fluid dynamics. This study is to find simplified methods that can be used to describe flow and thermal information from eight commonly used diffusers. The investigation used the box and momentum methods. The corresponding experimental data of airflow and thermal distributions from an environmental chamber have been used to validate the numerical methods. The box method works for most of the diffusers with an appropriate box size. The momentum method performs well for five diffusers. Since the momentum method is simpler than the box method, the momentum method should be used, whenever it is applicable.

Keywords: Air distribution, environmental control, indoor air quality, outlet, simulation, ventilation

INTRODUCTION

Computational fluid dynamics (CFD) is becoming a popular tool for indoor environment design. CFD modeling can greatly improve our knowledge of the performance of ventilation systems. Unfortunately, a major bottleneck to this modeling is that most diffusers are small compared to the size of the room and have high flow velocities. The requirements for modeling the flow inside a diffuser to predict the outflow characteristics are much different, and often incompatible, with the requirements for modeling the corresponding room airflow.

On the other hand, to model the room airflow, it is necessary to know the "boundary conditions" that describe the inlet jet airflow from a supply diffuser. The information that one typically can assume as an inflow boundary condition is usually limited to a profile in the inlet neck. On account of the complex geometries that air supply diffusers often have, actual modeling of the flow from the inside neck of the diffuser through and out of the diffuser is the only practical way of modeling the flow. Though modeling the flow like this might be theoretically possible, the codes that are efficient and available for modeling the room airflows are not capable of modeling the airflow in the diffuser. It is not clear which conditions must be specified, and where, in

* Jelena Srebric was a Research Assistant and Qingyan (Yan) Chen is an Associate Professor, Building Technology Program, Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA. Srebric is now an Assistant Professor, Department of Architectural Engineering, Pennsylvania State University, University Park, PA.

order to model the airflow in a room with a reasonable specification of the inlet boundary conditions.

Most of the current room airflow simulations only use simple diffusers because of this problem. As a result, many researchers have identified modeling the airflow supplied from the diffuser as a major limiting factor in applying CFD to room airflow (Moser 1991, Zhang et al. 1992, Vogel et al. 1993, Regard et al. 1995, and Jacobsen and Nielsen 1993).

Several recent studies have focused on this issue and have suggested potential solutions to this problem. For example, Skovgard and Nielsen (1991) compared two techniques of modeling diffuser flow, including modeling the diffuser directly, and modeling the resulting flow pattern in a volume in front of the diffuser (box method). They concluded that the box method had better performance, however, this depends on the diffuser-specific data. More recently, Emvin and Davidson (1996) discussed four methods of representing a diffuser with multiple jets. They concluded that further work is needed to develop a simplified inlet model without performing any measurements or sacrificing accuracy. Huo et al. (1996) also proposed a new method to describe diffuser boundary conditions. This method takes advantage of existing diffuser characteristic equations and the manufacturer's data. They concluded that the method may be used to accurately specify diffuser boundary conditions without describing the complicated diffuser geometry, and therefore, saving simulation time by using a coarser grid. Other recent reports on diffuser studies include work from Chen and Jiang (1996), Heikkinen and Piira (1994), and Joubert et al. (1996). Unfortunately, many of the above-mentioned studies lead to contradictory conclusions. Most of the studies have been limited to one or two diffusers.

This project builds on the work cited above by focusing on the identification of a simple and practical method that can be used by CFD modelers (ranging from designers to researchers) to model the airflow in a room supplied by a variety of real air supply devices. This investigation seeks a general method that can be applied to most commercial CFD codes.

RESEARCH APPROACH

Through an extensive literature review (Srebric 2000, Chen and Srebric 2000), we found that it is not possible to use jet formulae to provide boundary conditions of air supply diffusers for CFD modeling of room airflow. The review concludes that the simplified methods, such as Skovgard and Nielsen (1991) and Chen and Moser (1991), are promising to model diffuser airflow using resultant momentum. These methods provide the boundary conditions from the supply diffusers without modeling the detailed diffuser geometry. The methods can be grouped (Fan 1995) in two basic categories:

- Momentum modeling at the air supply devices, such as Chen and Moser (1991)
- Momentum modeling in front of the air supply devices, such as Skovgard and Nielsen (1991)

The momentum modeling approach at the air supply device imposes initial jet momentum as a boundary condition for CFD simulations. This approach is relatively easy to use, because the initial momentum is normally available. On the other hand, the approach for momentum modeling in front of the air supply device uses the momentum downstream from the diffuser. This approach determines the momentum in front of the

air supply device by using jet formulae or measured data. The jet formulae can be applied only in some special cases, such as fully developed free or attached diffuser jets, assuming that formulae are available for those types of diffusers. Unfortunately, for many practical applications, jets are neither free nor attached, and the transition region tends to be large. Furthermore, the main region of the jet is already being influenced by room airflow. Therefore, application of the momentum modeling in front of the diffuser requires measurements in many cases. However, not every diffuser manufacturer provides sufficient data for their product. Therefore, it is not easy to use the momentum modeling in front of the air supply devices.

Among many methods of momentum modeling at the air supply devices developed through an international effort (IEA 1993), the momentum method (Chen and Moser 1991) seems most promising. This method de-couples momentum and mass boundary conditions for the diffuser in a CFD simulation. The diffuser is represented in the CFD study with an opening that has the same gross area, mass inflow, and momentum flux as a real diffuser. This model enables specification of the source terms in the conservation equations over the real diffuser area. The air supply velocity for the momentum source term is calculated from the mass flow rate, \dot{m} , and the diffuser effective area A_o :

$$U_o = \frac{\dot{m}}{\rho A_o} \quad (1)$$

The momentum method requires the following data when it is used in a CFD simulation:

- airflow rate
- discharge jet velocity or effective diffuser area
- supply air turbulence properties
- supply air temperature and contaminate concentrations

The approach for momentum modeling in front of the air supply diffuser has also been used in many studies. Its variations include the box model (Nielsen 1989, 1997), the prescribed velocity model (Nielsen 1989), and the diffuser specification model (Huo et al. 1996). With a careful comparison (Srebric 2000, Chen and Srebric 2000), it seems that the box method was the most appropriate.

The box model (Nielsen 1989, 1997) represents the diffuser boundary conditions on an imaginary box surface around the diffuser. The flow field within the box is ignored. An example of the box method shown in Figure 1 illustrates how the method imposes boundary conditions for a nozzle diffuser in a two-dimensional form. One of the box surfaces (the front surface) uses jet profiles as boundary conditions, while the others use a free boundary with zero gradients for flow parameters. To specify the boundary conditions, suitable jet formulae should be applied. Or, air velocity, temperature and gas concentration profiles must be measured. Nielsen (1997) used the box method for a diffuser and obtained results in good agreement with the measured data. However, another study (Heikkinen 1991) found that the box method over-predicted the maximum jet velocity more than other simplified methods. Another problem with the box model is how one is to determine the box size. The box should be sufficiently large to have the boundaries in the fully developed jet region where the velocity and temperature profiles are similar. On the other hand, the box has to be small enough to avoid the impact of room air recirculation and thermal plumes on the jet.

In a CFD simulation, the box method needs information on:

- the distribution of air velocities
- the turbulence properties
- the distributions of temperature and contaminant concentrations

The box method seems to have a more solid physical background than the momentum method. However, it requires much more time for imposing the boundary conditions. The box method also needs some measured data for all the parameters simulated. Therefore, the momentum method is preferred, as it is easier to implement in a CFD simulation. Nevertheless, this investigation uses both methods to examine their performance in room airflow prediction.

For room airflow simulation, the present investigation used the CFD technique to solve a set of partial differential equations for the conservation of mass, momentum, energy, and species concentrations. These equations govern flow, heat, and mass transport in a room. Since airflow in the room is turbulent, the CFD technique used a turbulence model (the renormalized-group k- ϵ model from Yakhot et al. (1992)) to reduce the computing costs. With the turbulence model, the airflow, temperature, and species concentration transport can be described by the following unsteady time-averaged Navier-Stokes equations:

$$\frac{\partial(\rho\Phi)}{\partial t} + \text{div}(\rho V\Phi - \Gamma_{\phi,\text{eff}} \cdot \text{grad}\Phi) = S_{\phi} \quad (2)$$

Many textbooks have detailed the CFD theory, such as Versteeg and Malalasekera (1995). The governing equations can be closed with appropriate thermo-fluid boundary conditions at all the boundaries such as air inlets, outlets and wall surfaces. The values of velocity, temperature, kinetic energy, the dissipation rate of kinetic energy, and species concentration were set at the boundaries.

A commercial CFD program (CHAM 1998) was used in the present investigation to solve the time-dependent conservation equations together with the corresponding boundary conditions. The program discretized the indoor space into non-uniform computational cells, and the discrete equations were solved with the SIMPLE algorithm (Patankar 1980). The computed room airflows are validated with the measured data obtained from a full-scale environmental chamber (Yuan et al. 1999) and from the literature for a number of classic cases (Chen 1995).

SIMULATIONS OF COMMONLY USED AIR SUPPLY DIFFUSERS

In order to test the momentum and box methods, this investigation tested eight different diffusers commonly used in U.S. buildings. Different types of diffusers generate different airflow distributions, and may have fundamental differences in the jet flow structure. The eight different diffusers are: nozzle diffuser, slot (linear) diffuser, valve diffuser, displacement diffuser, round ceiling diffuser, square ceiling diffuser, vortex diffuser, and grille diffuser, as shown in Figure 2.

Figure 3 shows how the diffusers were installed in an environmental chamber, which is 17 ft (5.16 m) long, 12 ft (3.65 m) wide, and 8 ft (2.43 m) high. The nozzle and valve diffusers were not installed in the chamber, because experimental data for those diffusers were available from the literature. In addition to the internal heat gains as shown

in Table 1, there is 341 Btu/h (100 W) to 580 Btu/h (170 W) of cooling load from the window that depends on the diffuser type and ventilation rate. The ventilation rate studied ranges from 3.0 to 9.2 ACH. A more detailed description of the experimental setup is available from Yuan et al. (1999), Srebric (2000), and Chen and Srebric (2000).

The experiment measured the distributions of air velocity, air temperature, and the tracer gas (SF_6) concentration that was used to simulate a contaminant. The major measuring equipment includes: a flow visualization system by smoke for observing the airflow patterns, a hot-sphere anemometer system for measuring the air velocity and temperature, a thermocouple system for measuring the wall and air temperatures, and a tracer-gas system for measuring concentration. ASHRAE Standard 55 (1992) specifies the low-velocity range to be between 0.05 and 0.5 m/s (10 and 100 fpm) and requires an accuracy for the mean velocity measurements of ± 0.05 m/s (± 10 fpm). Our anemometers, omni-directional hot-sphere anemometers, were calibrated for a velocity as low as 0.05 m/s (10 fpm). However, the data is more reliable for a velocity higher than 0.1 m/s (20 fpm). The measuring errors for air temperature were ± 0.8 °F (± 0.4 K), including the errors introduced by the data acquisition systems. The wall and air temperature measurements with the thermocouples and data logger system have the same errors as the anemometers. The tracer gas was used to simulate the contaminants released from the two occupants at their head level. The gas detection threshold was around 10^{-3} ppm that should provide a very good accuracy for the present investigation.

Due to the limited space available in a technical paper, this paper presents only the results of the slot diffuser and the square ceiling diffuser in detail, since the two diffusers are very widely used. In addition, their flow characteristics are different. They are good representatives of the eight diffusers studied. However, the other diffusers will be discussed in order to draw general conclusions.

Slot (Linear) Diffuser

Figure 3 shows the installation position of the slot diffuser and the chamber layout used for the CFD simulations. The number of control volumes used for the simulations were $44 \times 26 \times 24$, $59 \times 38 \times 33$, and $72 \times 53 \times 44$. The grid independence was found with a grid resolution of $59 \times 38 \times 33$ and finer. Therefore, this paper uses the results with a grid resolution of $59 \times 38 \times 33$. This grid number is comparable to that used for a room without diffusers, such as natural convection. This also implies that the method to simulate the diffuser does not increase the computing time. Therefore, this method provides good grid economy and is computationally efficient.

Figure 2(e) shows the slot (linear) diffuser. The diffuser has openings that discharge parallel plane jets. The jets merged at a short distance from the diffuser because of the small separation distance ($3/4$ in. or 0.02 m) between the openings. According to the manufacturer's catalogue, the jet will be discharged horizontally without separation, as shown in Figure 4. Our smoke visualization illustrated that the jet was about 45° downwards to the rear wall ($x = 0$ plane in Figure 3) before it was reattached to the ceiling at 8 in. (0.2 m) from the diffuser. Since the jet attached to the upper sidewall as well ($y = 0$ plane in Figure 3), the overall jet flow was towards the upper right corner of the room.

Both the momentum and box methods were used to simulate the diffuser. The momentum method set the initial jet at 45° downwards over an area of 4 in. \times 46 in. (0.1 m \times 1.15 m), as shown in Figure 5(a).

With the box method, the measured data close to the diffuser were used to specify the boundary conditions in the CFD simulation, since no jet formulae are available for this diffuser. This study has used two types of box methods. The first method uses the measured data of the air temperature and velocity at 14 in. (0.36 m) downstream from the diffuser, as shown in Figure 5(b). At this position, the jets have merged into attached one. The box height shown in Figure 5(b) covers the downstream section, because our measurements in the opposite direction found very low air velocities (an average of 60 fpm or 0.3m/s) at 2 in. (0.06m) underneath the diffuser. Obviously, the low velocity is due to jet entrainment. The experiment measured the air velocity and temperature at 15 points on the box front surface. The data were used directly as the boundary conditions for the CFD simulation. The other box surfaces were modeled as zero-pressure, zero-gradients boundaries.

One major problem associated with the box method is how to specify the SF₆ concentration at the box surface. The tracer gas distribution was not uniform in front of the diffuser. It would take a long time to measure the SF₆ concentration at the 15 points where the air velocity and temperature were measured. As a result, this method is unlikely to be practical. Therefore, we did not simulate the tracer gas concentration with the box method.

This investigation also used a tiny box for the slot diffuser. The box is so small that the temperature and tracer-gas concentration at the box surface can be estimated without measurements from the energy and species balance. The average air velocity over the entire supply surface is used as the boundary condition for the CFD modeling. Our another paper (Srebric and Chen 2001a) details how to determine the box size and the corresponding flow and thermal boundary conditions with the tiny box method. With this method, if the peripheral jet velocities at the front surface of the box are lower than 80 fpm (0.4 m/s), they are neglected. Figure 5(c) shows the dimension of the tiny box. Although the size of the two boxes is very close, they are very different. The tiny box does not contain any recirculated flow so that the boundary conditions for temperature and SF₆ concentration can be easily determined, according to the method described in Srebric and Chen (2001a).

Figures 6, 7, and 8 present the calculated results of the air velocities, air temperatures, and SF₆ concentrations in the room, respectively. The corresponding experimental data are used in the figures for the validation of the numerical results.

The calculated air velocities agree reasonably with the measured data, as shown in Figure 6. It seems that the discrepancies between the computed and measured results for poles 4 and 5 are larger than those shown in the other poles. In fact, the discrepancies are the same, because both poles 4 and 5 use a smaller coordinate scale. Most of the discrepancies are within the error caused by the omni-directional anemometers (10 fpm or 0.05 m/s). The computed velocity profiles with the box method and the tiny box method could not show correct results on the upper part of pole 3. This is because pole 3 is located within the box, and so no computed data are available. Therefore, zero velocity is plotted in the figure, while the measurements and the momentum method show a high velocity.

The momentum method failed to predict the jet development. The jet decay predicted is too fast, as shown in the upper part of poles 1 and 2. The results are much worse if the grid number is reduced. In many cases, the jet may drop earlier than the

designed value in a variable air volume system. Correct prediction of the jet is crucial for thermal comfort design. Therefore, the performance of the momentum method is not satisfactory in predicting the air velocity. The exact reason for the error in the momentum method is not clear.

The tiny box method could not predict the jet flow due to the flat velocity profile used at the supply surface. Figure 6 clearly shows that the flat jet velocity profile occurs at pole 2, which is already 1.5 ft (0.45 m) in front of the box. The flat velocity profile also has a significant impact on the calculated jet temperatures (Figure 7). To improve the results, the measured velocity profiles should be used in the tiny box method. Of course, the effort in that direction would cost more time and labor because of the need of more data. Nevertheless, the balance treatment for the temperature and concentration can be used in the tiny box method to predict a reasonably good concentration profile, as shown in Figure 8.

Figure 7 shows the computed and measured temperature profiles. The computed profiles are in agreement with the data in recirculation region. However, the performance in predicting the temperature in the jet region is very different. The momentum method and the tiny box method predicted a too low air temperature profile (see poles 1 and 2 in Figure 7). The jet temperatures were under-predicted because the calculated jet velocities were higher than the measured ones that implies a poorer mixing with the surrounding air. The results with the tiny box method may be improved if a velocity profile at the box surface is used instead of a single value. Only by using the box method with the velocity profile, which was simulated by 15 patches, one can accurately describe the boundary conditions for this diffuser. Clearly, the box method performs better than the momentum method in this case.

On account of the difficulties in specifying the boundary conditions for the box method, only the momentum and the tiny box method were used to calculate the tracer-gas concentration profiles (see Figure 8). With the momentum method, the supply boundary condition (such as the zero concentration) from the diffuser can be easily specified at the inlet. The tiny method can use a zero-concentration, because the flow at the box surface has not mixed with the room air. However, the zero-concentration assumption cannot be used in the box method, because some room air has mixed with the inflow at the box surface.

It is interesting to note that even the sharp gradient of the tracer-gas in pole 1 is well predicted (See Figure 8), where the tracer gas source is closely located. Normally, a point source creates a locally high gradient of tracer gas concentration that is difficult to predict numerically. Unfortunately, the momentum method failed to predict the tracer-gas concentration at pole 2 because the method over-predicts the convection around the tracer gas source due to the over-predicted jet momentum. The performance of the tiny box method was better.

The results from the momentum method, the box method, and the tiny box method conclude that the accuracy in simulation boundary conditions for a complex diffuser is proportional to the labor in describing the diffuser. There is always a trade-off between accuracy and labor for this particular diffuser. For better accuracy, the box method or the tiny box method with supply velocity profile should be used for the slot diffuser.

Square Ceiling Diffuser

The square ceiling diffuser discharges jets in four cross and four diagonal directions, as shown in Figure 9(a). The simulation of the square diffuser used both the momentum and the box methods. Figure 9(b) shows schematically how the flow direction needs to be described for the diffuser in the momentum method, and Figure 9(c) shows the box size used in the box method for the diffuser. The box method requires a high labor effort because it specifies flow information in four surfaces, while the momentum method requires only the flow information on one surface.

Since the jets from the diffuser are very thin (around 2 in. or 0.05 m), the measurements are more difficult to take due to the sharp velocity gradient. In this case, the box method used 40 measured velocities and a uniform temperature obtained from the heat balance for the box boundaries. On the other hand, the momentum method used a discharge velocity for the entire diffuser surface as the momentum source. The discharge velocity is an averaged maximum velocity measured at the perimeter of the diffuser. The flow direction specified in the momentum method was obtained from the smoke visualization. The air temperature used was the same as the supply air temperature in the duct.

The calculations used a $48 \times 40 \times 33$ grid distribution that is grid independent. Again, this grid resolution is comparable for a case without diffuser so that it is computationally efficient. Figure 10 compares the measured and calculated air velocities with the two methods at five positions in the room. The right bottom figure shows the floor plan. It seems that the two methods gave similar results, although the momentum method performed better in areas near poles 4 and 5, which were close to the diffuser. Since the velocity was measured with omni-directional anemometers, the anemometers have a great uncertainty in measuring low air velocity, because the natural convection from the heated probes would generate a false velocity of the same magnitude. The estimated measuring error is also shown in the figure.

It is interesting to note that the box method over-predicted the air velocity along the jet region, although the measured data was used to specify the boundary conditions. The reason is not clear. On account of this over-predicted velocity in the jet region, the corresponding temperature predicted with the box method is lower than the measured one, as shown in Figure 11. However, the two methods can correctly predict the air temperature in the occupied zone. This is not surprising, since the room airflow is well mixed, and the energy balance ensures a correct mean air temperature.

This investigation has also validated the predicted tracer gas concentration by the experimental data is illustrated in Figure 12. The computed results are only for the momentum method. The results show a uniform distribution of the trace-gas concentration in the entire chamber, and the agreement between the computed results and the experimental data is good. The box method can also be used to calculate the tracer gas concentration distribution with species balance, because the box used is very small. Since it is a well-mixed condition, a similar result is anticipated, as indicated in the temperature prediction.

The results show that both the momentum and box methods predicted good results for the square ceiling diffuser. Since it is simpler to specify the boundary conditions, the momentum method is recommended.

Furthermore, this study has also used another square ceiling diffuser from a different manufacturer to measure air velocity and air temperature. The purpose of the added study is to identify whether the results obtained for the first diffuser are universal. The results, although not shown here, indicate that the difference in room air velocity is within the measuring errors. The two temperature fields are nearly identical.

DISCUSSION

On account of the limited space, the paper presented only slot (linear) and square ceiling diffusers, although eight different kinds of diffusers have been investigated. The slot and square ceiling diffusers produce well mixed flow field in the entire chamber that results in uniform velocity, temperature and tracer-gas concentration distributions. The simplified methods for diffuser simulations were able to predict the main properties of the flow, temperature and contaminant concentration fields. However, the accuracy of the simplified methods is a function of the amount of data incorporated in the modeling. Two simplified methods for diffuser modeling, the box and momentum methods, were tested.

The box method that requires measured resultant velocities, temperatures and concentration distribution at the box surface, is the most accurate, and probably not very practical. For example, the square ceiling diffuser has attached ceiling jets that are very thin (about 2 in. or 0.05 m), and the gradients for all air parameters in the jet region are very large. Therefore, taking accurate measurements to obtain the data for the box method are very tedious, and the flow directions may not be as easily determined with the smoke visualization due to the high velocities. Furthermore, the specification of the box boundary conditions is time consuming, since the jets spread in all radial directions from the square diffuser.

Our study developed a modified version of the box method called the tiny box method that uses a relatively small box and a measured resultant velocity field to specify the diffuser boundary conditions. The boundary conditions for supply temperature and contaminant concentration are calculated from energy and species balance for the tiny box (Srebric and Chen 2001a). The tiny box method produces similar results as the original box method, while using a minimal amount of measured data.

The momentum method that requires only a discharge velocity or effective diffuser area, sometimes available in the manufacturer catalogues, is the simplest one, and produced good results for the square diffuser. Therefore, the momentum method is preferred and recommended over the box method for this diffuser. However, if the jet development in front of a diffuser is complex, such as in the case of slot diffuser, then the momentum method is not sufficiently accurate and the box method is recommended.

Table 2 shows the recommended methods for each of the eight different diffusers we have studied. It is possible to classify diffusers into three groups: diffusers to use only the momentum method, diffusers to use only the box method, and diffusers to use both methods.

Diffusers to Use Only the Momentum Method

The table shows that only the momentum method can be used for the displacement diffuser. Yuan et al. (1999) have used the momentum method to predict airflow, air temperature, and contaminant concentration in a room with the displacement diffuser. The results are in very good agreement with the experimental data. Figure 13

presents the computed and measured velocity profile in the middle of the room for the present study also available in another paper (Srebric and Chen 2001b). The displacement diffuser produces stratified distributions of air temperature and contaminant concentrations. The agreement between measured and calculated temperatures is very good, even in the jet region. The prediction of the contaminant concentrations shows some discrepancies between measurements and calculations. The discrepancies are due to difficulties in simulating contaminant sources that are point sources and have an initial momentum.

The reason for the box method to fail is that the buoyancy force plays an important role in the jet development. The jet flow and air temperature change rapidly in the diffuser vicinity. The measurements for the jet development would have to be performed under non-isothermal conditions. For non-isothermal conditions, the data could only be used to simulate the diffuser in a room with the same thermal conditions as the measured ones. Any change in the room geometry and/or heat sources could have a significant influence on the data. The application would be very limited. Therefore, the box method is not recommended for the displacement diffuser.

Diffusers to Use Only the Box Method

For the nozzle, slot, and valve diffusers, the box method can simulate the boundary conditions better than the momentum method. This is because the flow mixing is too complex for the momentum method to handle. The flow develops, merges, and combines in front of the diffusers to form a developed jet. The momentum method may not be appropriate to represent the complexity of the flow field in front of the diffusers. To properly simulate this flow, a very fine grid resolution is required. Even though a very fine grid resolution may not provide a correct prediction of the airflow pattern, as in the case of the nozzle diffuser (Emvin and Davidson 1996). Furthermore, for the valve diffuser, the momentum method was very sensitive to the discharge angle that was not observed with the smoke visualization. Therefore, the box method is recommended to simulate these complex diffusers, as it is more stable and reliable.

Figure 13 shows the velocity profiles in the middle of a room predicted with the box method for the nozzle diffuser, in addition to the slot diffuser reported in Figure 6. The valve diffuser had only qualitative comparison between calculated and measured velocities and airflow pattern because the detailed measured data were available only in the vicinity of the diffuser. Please note that the present study used the experimental data from the literature for the nozzle and valve diffuser. The corresponding room size and thermal and flow conditions are different from those used for the other diffusers. The results shown in Figure 13 confirm that the box method is capable to provide satisfactory results for room air distribution design. Srebric (2000) and Chen and Srebric (2001) have provided more detailed validation of the nozzle and valve diffusers.

Diffusers to Use both Methods

Among the eight diffusers studied, both the momentum and box methods can be used to predict room airflow with the square ceiling, round ceiling, vortex, and grill diffusers. The round ceiling diffuser discharges a radial jet attached to the ceiling in a very thin layer (around 0.05 m or 2 in). The jet is similar to the one created by the square ceiling diffusers. One would anticipate the momentum and box methods would perform

the same for the round ceiling diffuser as for the square diffuser. Figure 13 illustrates the velocity profile in the middle of the room with the diffuser, predicted with the momentum method also available in another paper (Srebric and Chen 2001b). Figure 13 also shows the velocity profiles in the middle of the room for the vortex and grille diffusers. The results were obtained with the momentum method. Again, they agree with the experimental data. The calculated temperature field agreed very well with the measured data for all diffusers in this group. The calculations and measurements of the contaminant concentrations show discrepancies similar to the one in Figure 12. The probable cause of the discrepancies is the nature of the contaminant sources used for the experiments that are point sources with an initial momentum.

With the box method (Chen and Srebric 2001), the computed airflow agrees with the experimental data. The accuracy is similar to that with the momentum method. Although both methods are good, the momentum method is recommended because it is simpler.

The present study has limitations. Although the eight diffusers tested represent a wide variety of the diffusers available on the market, this investigation does not ensure that the results can be extended to a diffuser of similar kind. In some cases, cluster diffusers may be used for a large space. The impact of one diffuser on the other has not been examined. Furthermore, the jets from the diffusers are free from the interference of obstacles, such as beams and columns. The impact of the obstacles on the two methods needs to be further studied. In addition, the present investigation does not study the impact of turbulence intensity on airflow and how it should be modeled, although turbulence intensity is a key parameter for draft.

CONCLUSIONS

The objective of this research was to develop a simplified methodology to specify boundary conditions of air supply diffusers for CFD simulations. Two simplified methods, the momentum and box methods, have been validated with the measured data in order to assess their accuracy and reliability. The measured data were obtained from a state-of-the-art experimental facility, as well as from the literature.

Our conclusions with modeling diffuser jet flow can be summarized as follows:

- Numerical simulations of the diffusers can employ a simplified modeling technique by using the resultant momentum from the diffusers, without the detailed diffuser geometry representation. The modeling of the resultant momentum can be made at the diffuser surface with the momentum method, or in front of the diffuser in the direction of the jet discharge with the box method. The momentum method is simpler than the box method in terms of the effort to specify the data for modeling.
- The momentum method is recommended for the displacement diffuser and the mixing diffusers that discharge combined jets. The mixing diffusers, such as the nozzle, slot and valve diffusers, that discharge several jets that merge and combine in front of them should use the box method, since the momentum method performs poorly.
- With a proper box size, the data required for the box method, such as temperature and contaminant concentration distributions, can be eliminated, except the air velocity distribution.

- The simplified diffuser modeling requires experiments to measure some of the parameters necessary for the simulations or more detailed information from a product catalogue.

NOMENCLATURE

A_o	= diffuser effective area
\dot{m}	= mass flow rate
S_ϕ	= source term
T	= time
U_o	= air velocity specified in the CFD simulation
V	= velocity vector
$\Gamma_{\phi,eff}$	= effective diffusion coefficient
Φ	= 1 for mass continuity
	= V_j ($j = 1, 2, 3$) for three components of momentum
	= k for turbulent energy
	= ε for the dissipation rate of k
	= T for energy transport
	= C_i for contaminant concentration i
ρ	= air density

REFERENCES

- ASHRAE. 1992. "Thermal environmental conditions for human occupancy," ANSI/ASHRAE Standard 55 - 1992, ASHRAE, Atlanta.
- CHAM. 1998. "PHENICS Version 3.1", CHAM Ltd., London, UK.
- Chen, Q. 1995. "Comparison of different k - ε models for indoor airflow computations," Numerical Heat Transfer, Part B: Fundamentals, 28, 353-369.
- Chen, Q., and Jiang, Z. 1996. "Simulation of a complex air diffuser with CFD technique," Proc. of ROOMVENT '96, Vol. 1: 227-234.
- Chen, Q., and Moser, A. 1991. "Simulation of a multiple-nozzle diffuser," Proc. of 12th AIVC Conference, Vol. 2: 1-14.
- Chen, Q. and Srebric, J. 2000. "Simplified diffuser boundary conditions for numerical room airflow models," Final Report for ASHRAE RP-1009, 181 pages, Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA.
- Emvin, P. and Davidson L. 1996. "A numerical comparison of the diffuse in case E1 Annex20," Proc. of ROOMVENT '96, Vol. 1: 219-226.
- Fan, Y. 1995. "CFD modeling of the air and contaminant distribution in rooms," Energy and Buildings, 23: 33-39.
- Heikkinen, J. 1991. "Modeling of a supply air terminal for room air flow simulation," Proc. of 12th AIVC Conference, Vol. 3: 213-230.
- Heikkinen, J., and Piira, K. 1994. "CFD computation of jets from circular ceiling diffusers," Proc. of ROOMVENT '94, Vol. 1: 329-344.
- Huo, Y., Zhang, J., Shaw, C., and Haghghat, F. 1996. "A new method to describe the diffuser boundary conditions in CFD simulation," Proc. of ROOMVENT '96, Vol. 2: 233-240.

- IEA. 1993. "Annex 20: Room air and contaminant flow, evaluation of computational methods," Subtask-1, Summary report, ed. A. D. Lemaire, TNO Building and Construction Research, Delft.
- Jacobsen T.V. and Nielsen P.V. 1993. "Numerical modeling of thermal environment in a displacement-ventilated room" Proc. of Indoor Air 93, Vol. 5.
- Joubert, P., Sandu, A., Beghein, C. and Allard, F. 1996. "Numerical study of the influence of inlet boundary conditions on the air movement in a ventilated room," Proc. of ROOMVENT '96, Vol. 1: 235-242.
- Moser, A. 1991. "The Message of Annex 20: Air Flow Patterns within Buildings" Proc. 12th AIVC Conference.
- Nielsen, P.V. 1989. "Representation of boundary conditions at supply openings," IEA, Annex 20, Research Item 1.11.
- Nielsen, P.V. 1997. "The box method - a practical procedure for introduction of an air terminal device in CFD calculation," Institute for Bygningsteknik, Aalborg Universite, Denmark.
- Patankar, S.V. 1980. Numerical Heat Transfer and Fluid Flow, New York: Hemisphere Publishing Corp.
- Regard M., Carrie F.R., Voeltzel A, and Richalet V. 1995. "Measurement and CFD modeling of IAQ indices," Proc. of 16th AIVC Conference.
- Skovgaard, M., and Nielsen, P.V. 1991. "Modelling complex inlet geometries in CFD - Applied to air flow in ventilated rooms," Proc. of 12th AIVC Conference, Vol. 3: 183-200.
- Srebric, J. 2000. "Simplified methodology for indoor environment design," Ph.D. Thesis, Department of Architecture, Massachusetts Institute of Technology, Cambridge, MA.
- Srebric, J. and Chen, Q. 2001a. "A method of test to obtain diffuser data for CFD modeling of room airflow," ASHRAE Trans., 107(2). Srebric, J. and Chen, Q. 2001b. "Boundary conditions for diffusers in room air distribution calculations," Proceedings of CLIMA 2000, Napoli, Italy
- Versteeg, H.K. and Malalasekera, W. 1995. *An Introduction to Computational Fluid Dynamics*, London: Longman.
- Vogel P., Richter E. and Rosler M. 1993. "The effect of various inlet conditions on the flow pattern in ventilated rooms - measurements and computations", Proc. of 14th AIVC Conference.
- Yokhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., and Speziale, C.G. 1992. "Development of turbulence models for shear flows by a double expansion technique," Physics Fluids A, 4(7): 1510-1520.
- Yuan, X., Chen, Q., Glicksman, L.R., Hu, Y., and Yang, X. 1999. "Measurements and computations of room airflow with displacement ventilation," ASHRAE Trans., 105(1): 340-352.
- Zhang J.S., Christianson L.L., Wu G.J., and Zhang R.H. 1992. "An experimental evaluation of a numerical simulation model for predicting room air motion", Proc. of Jacques Cartier Conference.

Table 1. Internal heat sources in the environmental chamber

Internal Heat Sources	Btu/hr (W)
Each human simulator	256 (75)
Computer 1	368 (108)
Computer 2*	590 (173)
Each fluorescent lamp	116 (34)
TOTAL	1935 (567)

* The one close to the window

Table 2. Summary for the simplified diffuser modeling

Diffusers	Box Method	Momentum Method	Recommended Method
Nozzle	Good	Poor	Box
Slot (Linear)	Good	Poor	Box
Valve	Good	Poor	Box
Displacement	Poor	Good	Momentum
Square Ceiling	Good	Good	Momentum
Round Ceiling	Good	Good	Momentum
Vortex Ceiling	Good	Good	Momentum
Grille	Good	Good	Momentum

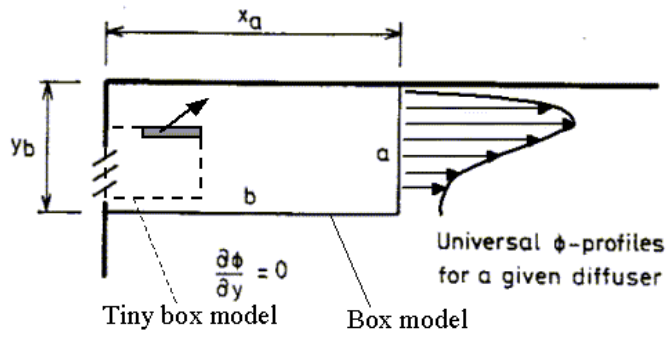
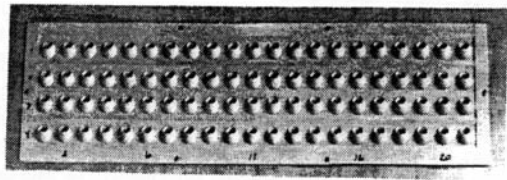


Figure 1. The box and tiny box model for a nozzle air supply diffuser



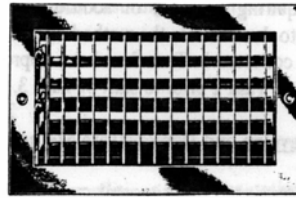
(a)



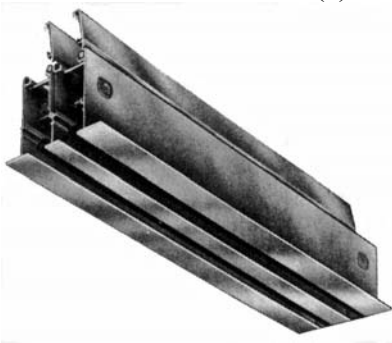
(b)



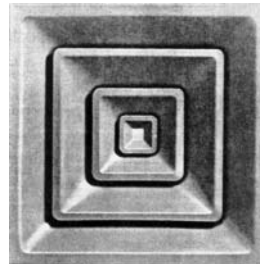
(c)



(d)



(e)



(f)



(g)



(h)

Figure 2. Eight commonly used air supply diffusers: (a) nozzle, (b) valve, (c) displacement, (d) grille, (e) slot, (f) square ceiling, (g) round ceiling, and (h) vortex.

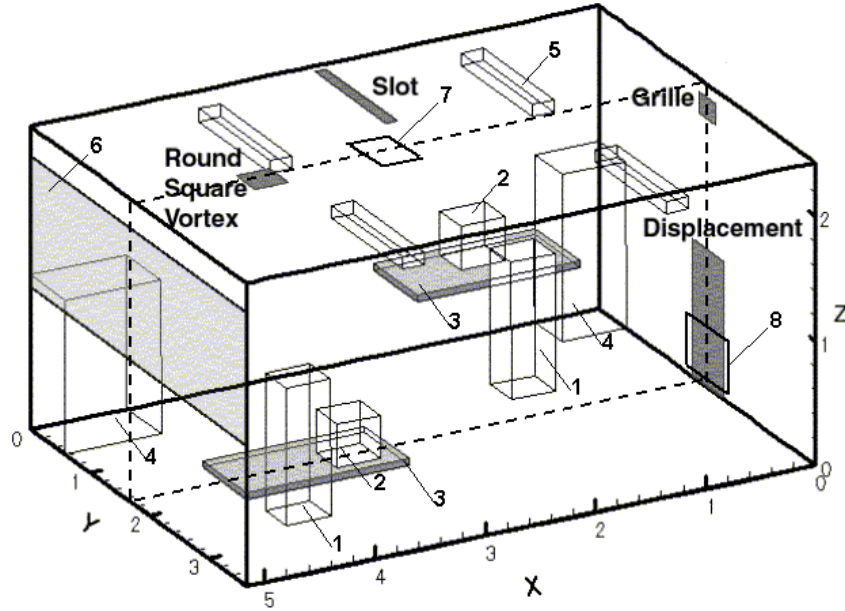


Figure 3. Positions of the supply diffusers in the test chamber (person - 1, computer - 2, table - 3, cabinet - 4, fluorescent lamp - 5, window - 6, exhaust for the displacement diffuser - 7, exhaust for the mixing diffusers - 8)

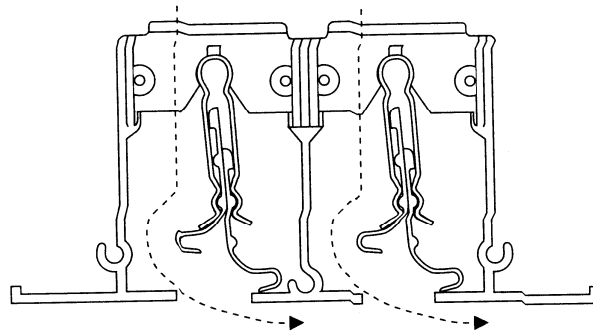


Figure 4. The airflow pattern from the slot diffuser manufacturer's catalogue

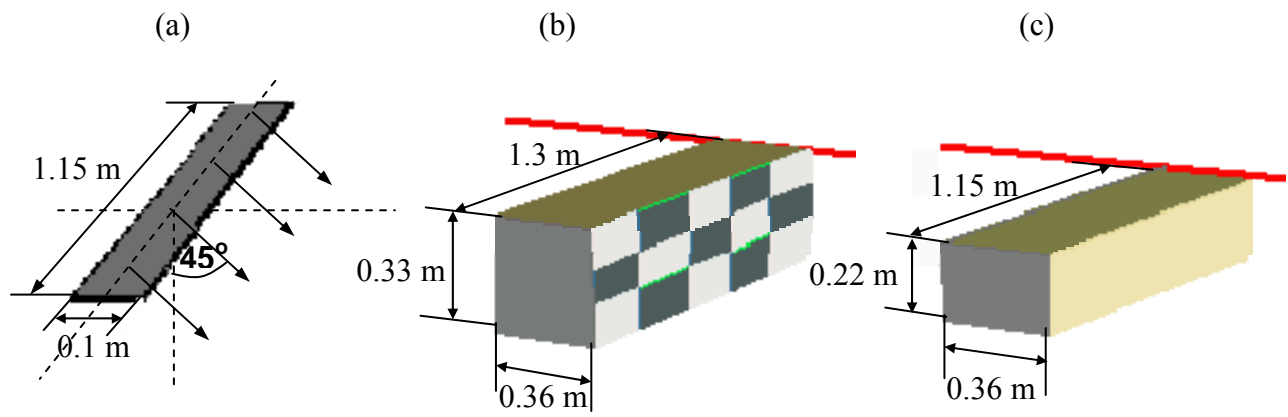


Figure 5. Simulation of the slot diffuser with (a) the momentum method (b) the box method, and (c) the tiny box method

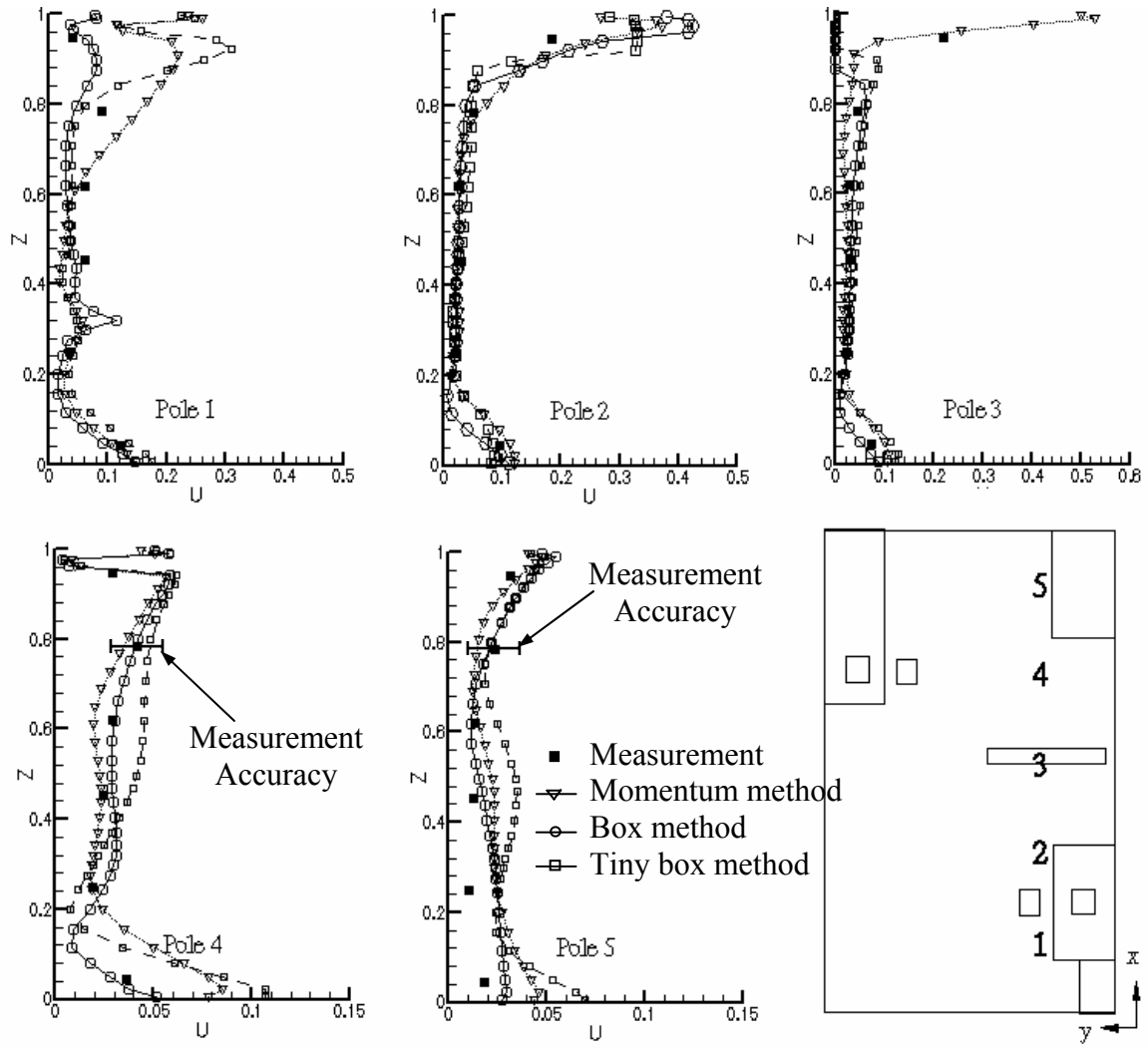


Figure 6. The comparison of the calculated and measured velocity profiles for the slot diffuser at five positions in the room Z =height/total room height (H), U =velocity/supply velocity (U_0), $H=2.43\text{m}$, $U_0=3.9\text{m/s}$

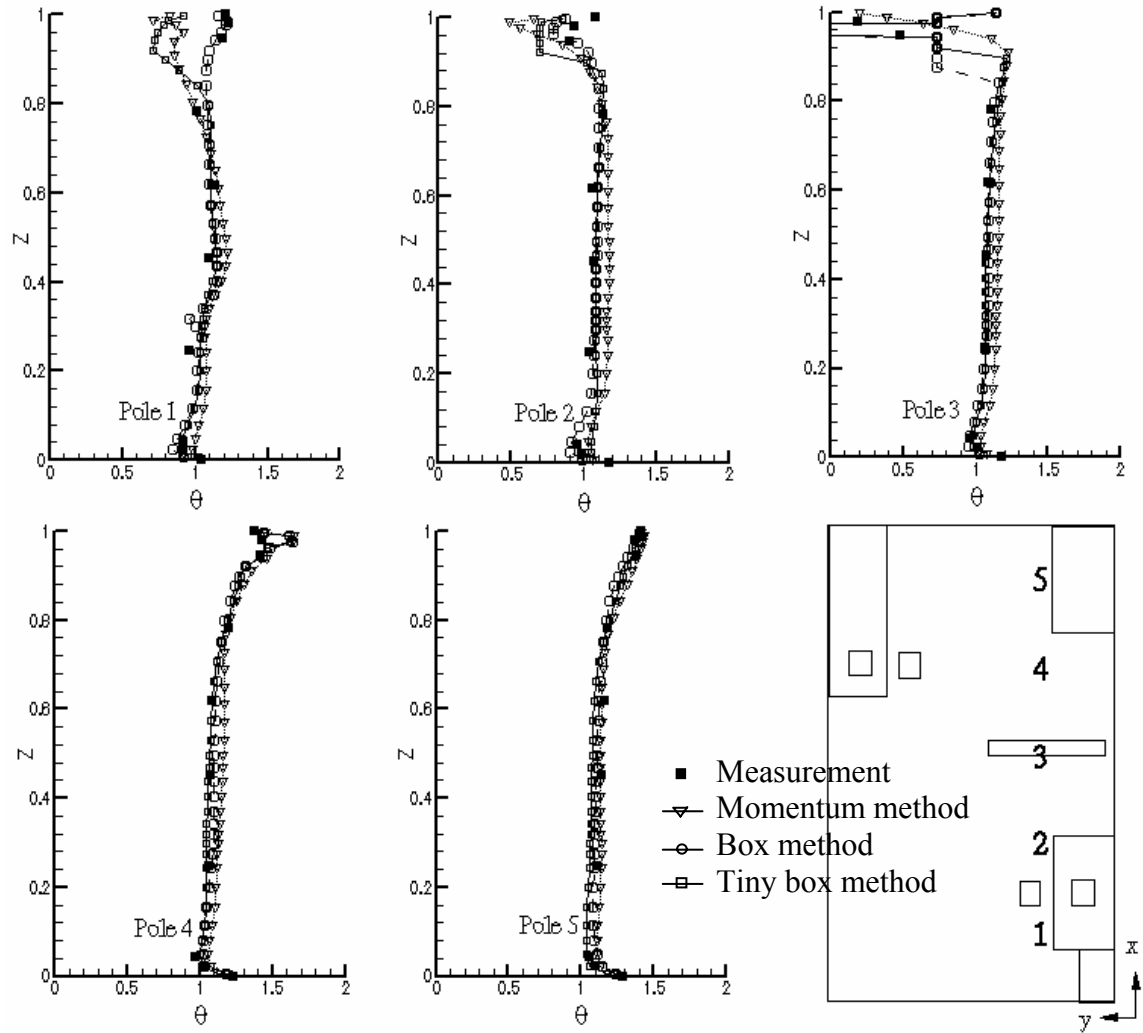


Figure 7. The comparison of the calculated and measured temperature profiles for the slot diffuser at five positions in the room, Z =height/total room height (H), $\theta=(T-T_{in}/T_{out}-T_{in})$, $H=2.43\text{m}$, supply air temperature $T_{in}=16.3^{\circ}\text{C}$, exhaust air temperature $T_{out}=21.4^{\circ}\text{C}$

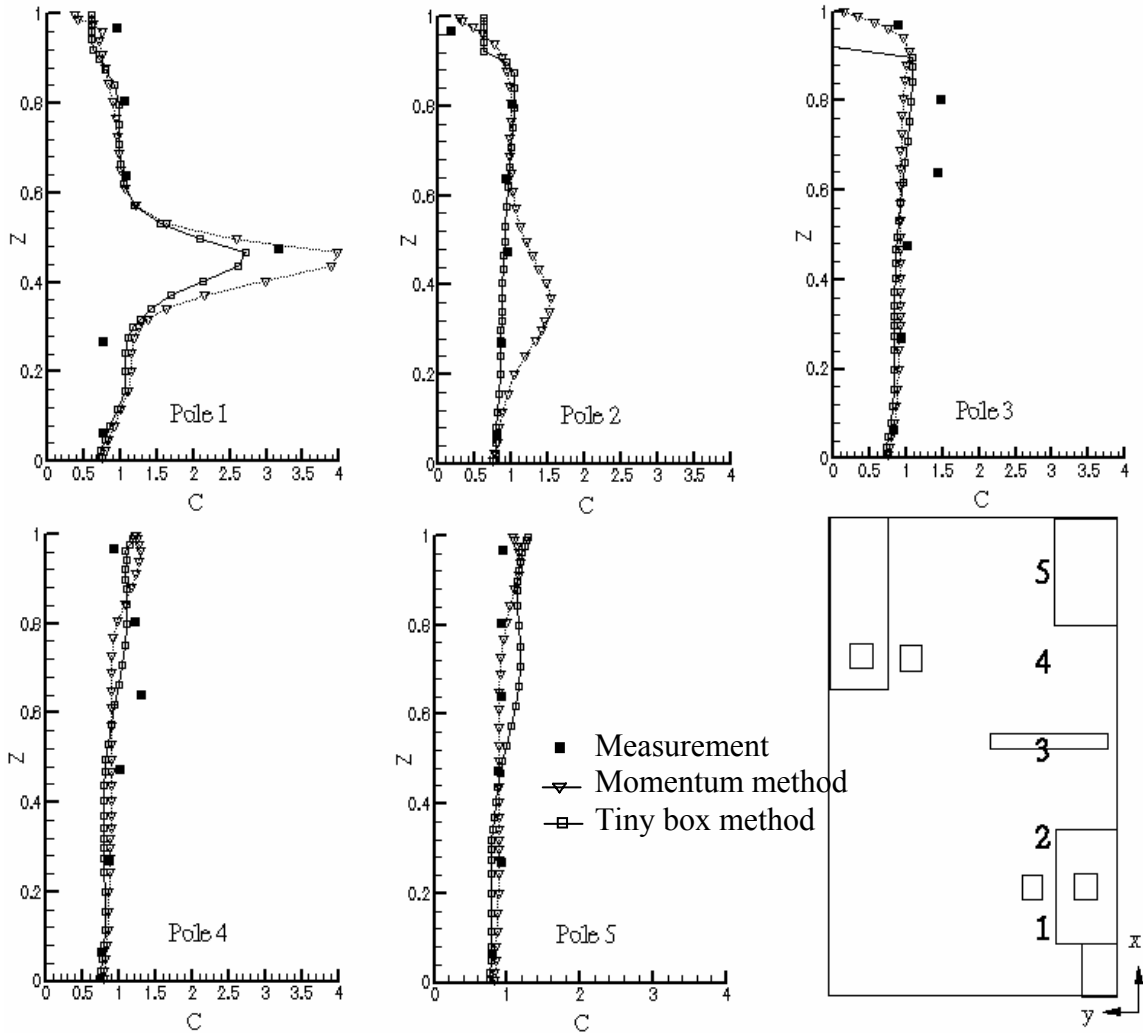
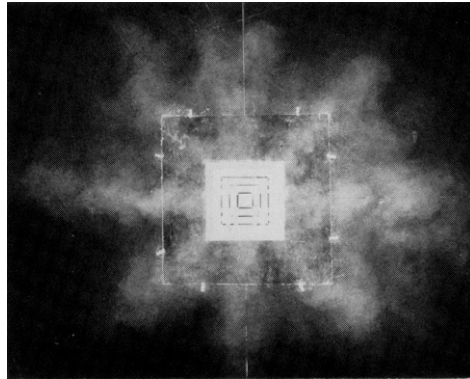
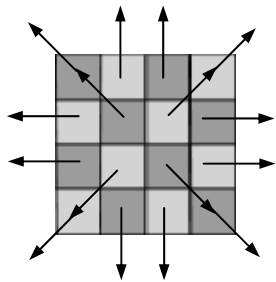


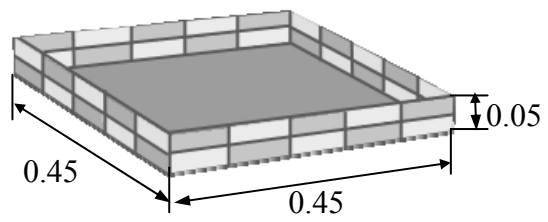
Figure 8. The comparison of the calculated and measured SF₆ concentration profiles for the slot diffuser at five positions in the room, $Z = \text{height}/\text{total room height (H)}$, $C = (c - c_{in}) / (c_{out} - c_{in})$, $H = 2.43\text{m}$, supply SF₆ concentration $c_{in} = 0.0327\text{ ppm}$, exhaust SF₆ concentration $c_{out} = 0.5546\text{ ppm}$



(a)



(b)



(c)

Figure 9. (a) Flow visualization for the square ceiling diffuser, (b) the velocity information needed for the momentum method, and (c) the box size for the box method

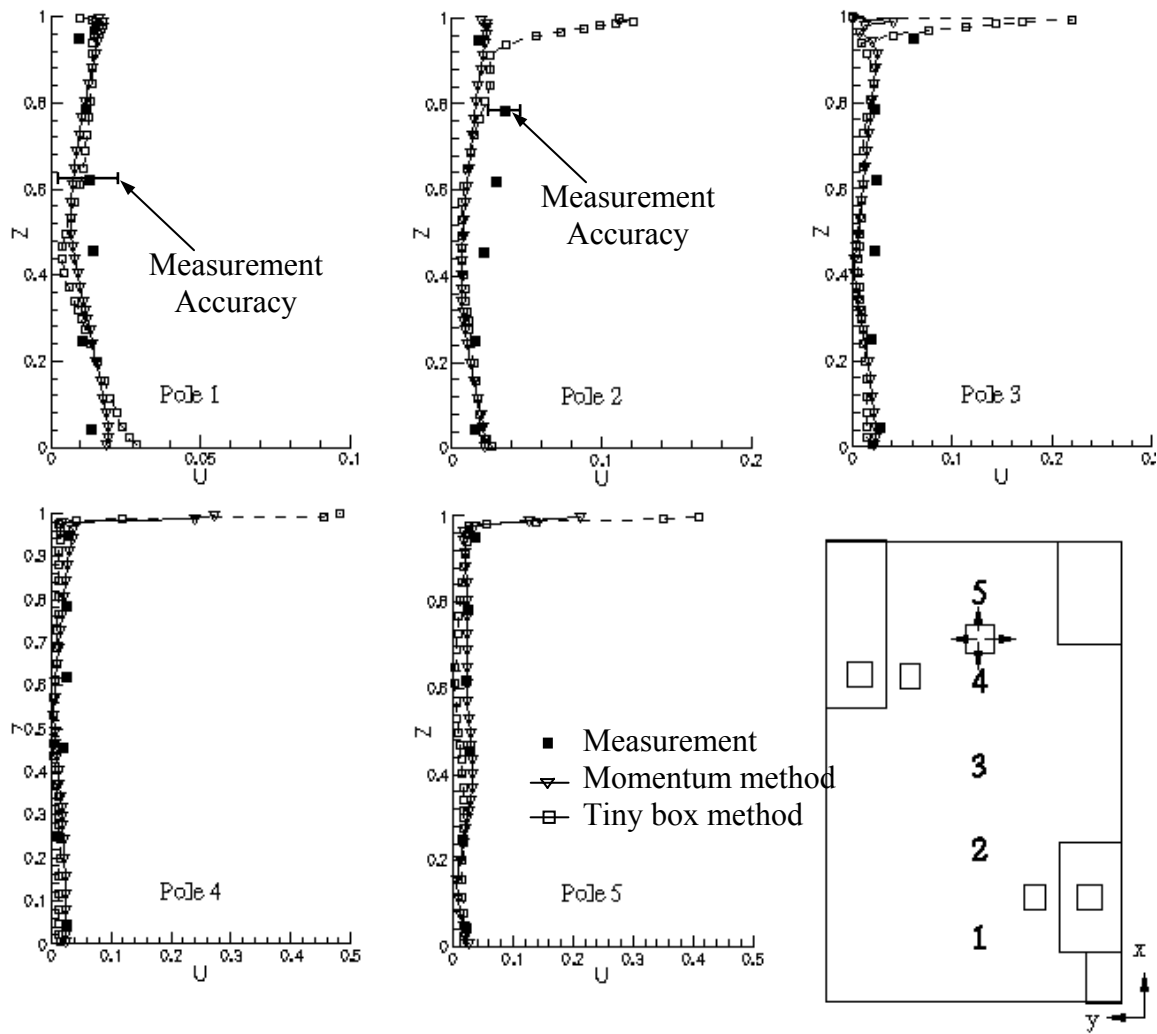


Figure 10. The comparison of the calculated and measured velocity profiles for the square ceiling diffuser at five positions in the room, Z =height/total room height (H), U =velocity/supply velocity (U_0), $H=2.43\text{m}$, $U_0=5.2\text{m/s}$

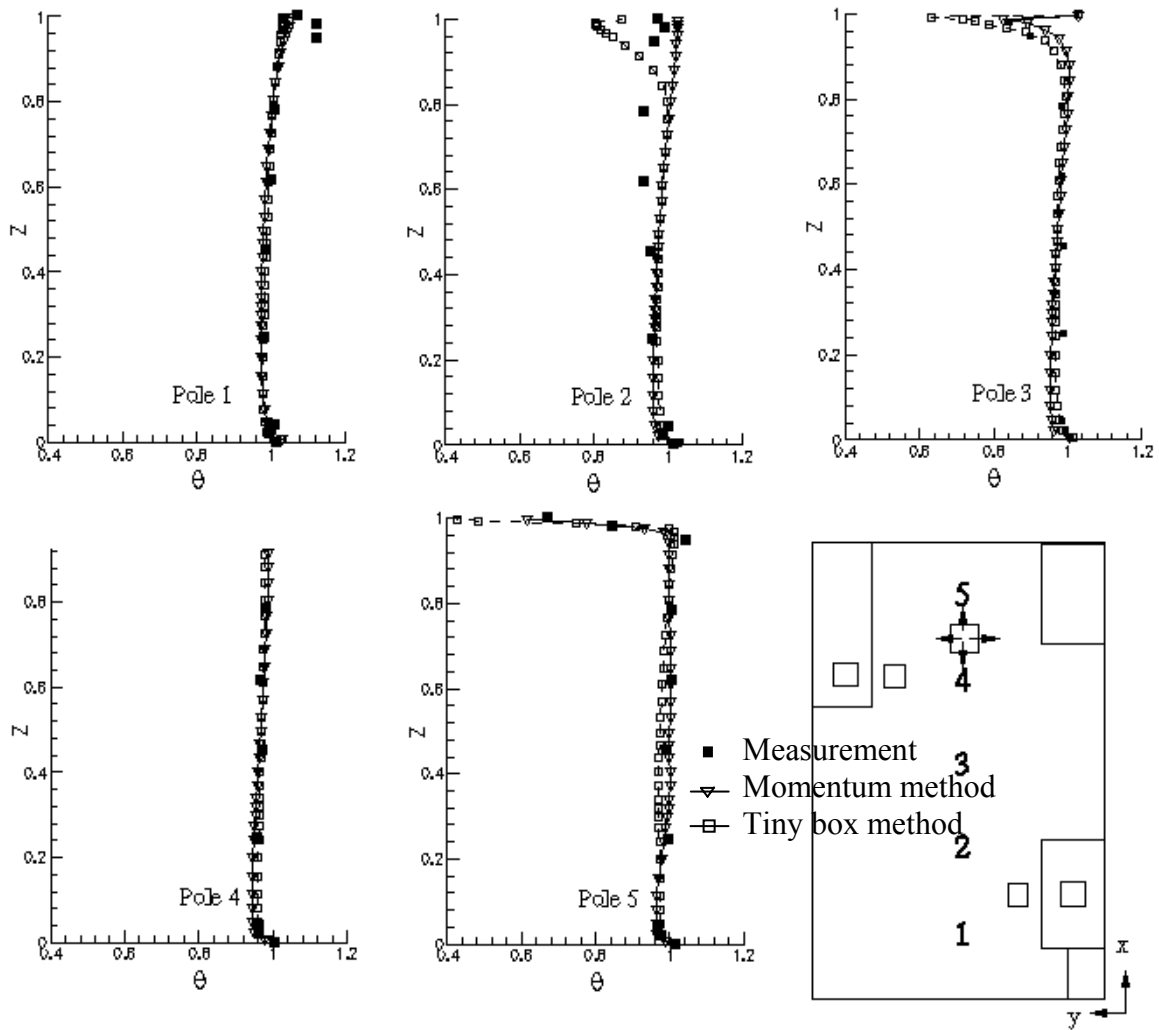


Figure 11. The comparison of the calculated and measured temperature profiles for the square ceiling diffuser at five positions in the room, Z =height/total room height (H), $\theta=(T-T_{in}/T_{out}-T_{in})$, $H=2.43\text{m}$, supply air temperature $T_{in}=14.5^{\circ}\text{C}$, exhaust air temperature $T_{out}=24.1^{\circ}\text{C}$

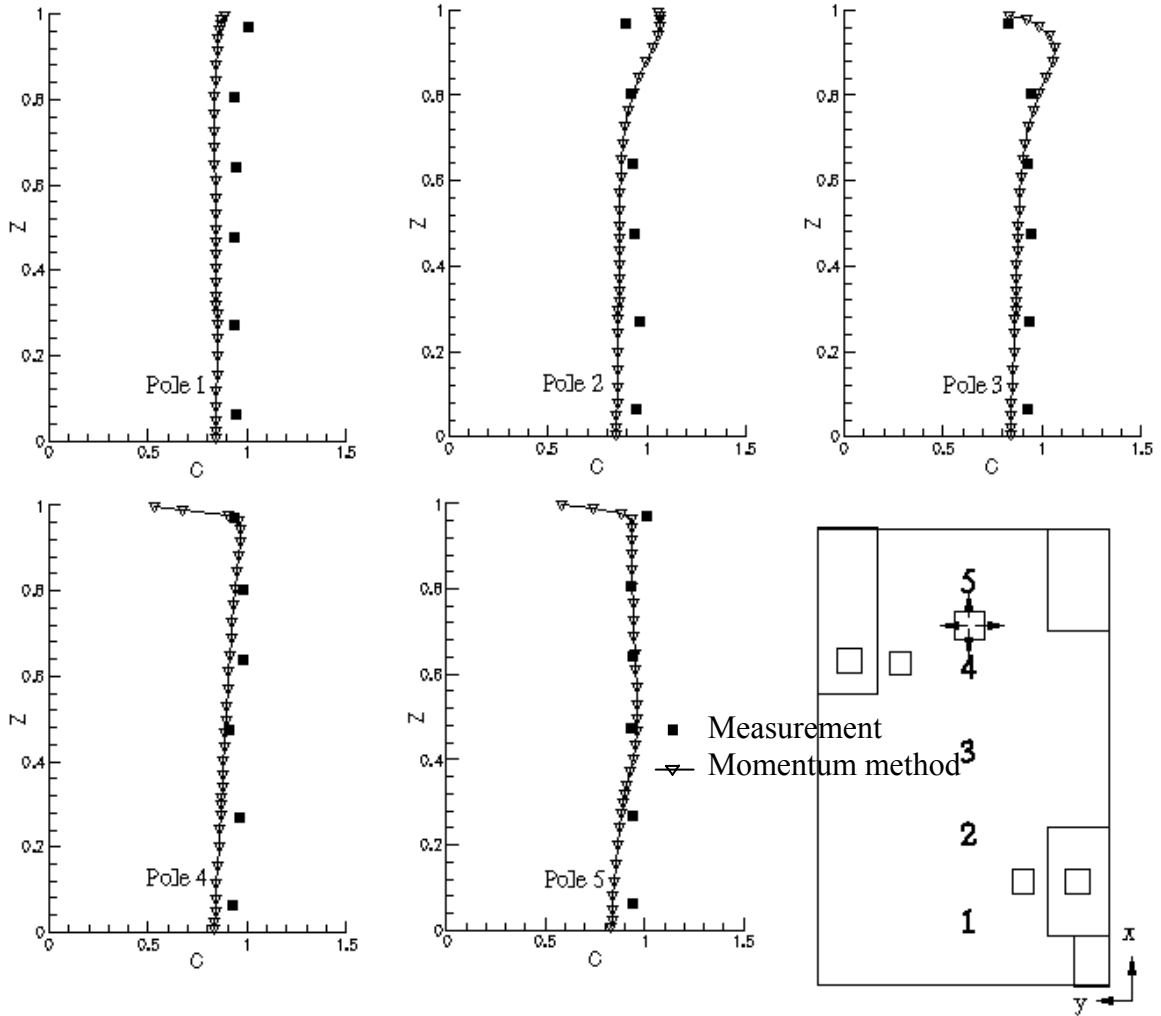


Figure 12. The comparison of the calculated and measured SF₆ concentration profiles for the square ceiling diffuser at five positions in the room, $Z = \text{height}/\text{total room height (H)}$, $C = (c - c_{in}) / (c_{out} - c_{in})$, $H = 2.43\text{m}$, supply SF₆ concentration $c_{in} = 0.0492\text{ ppm}$, exhaust SF₆ concentration $c_{out} = 1.0496\text{ ppm}$.

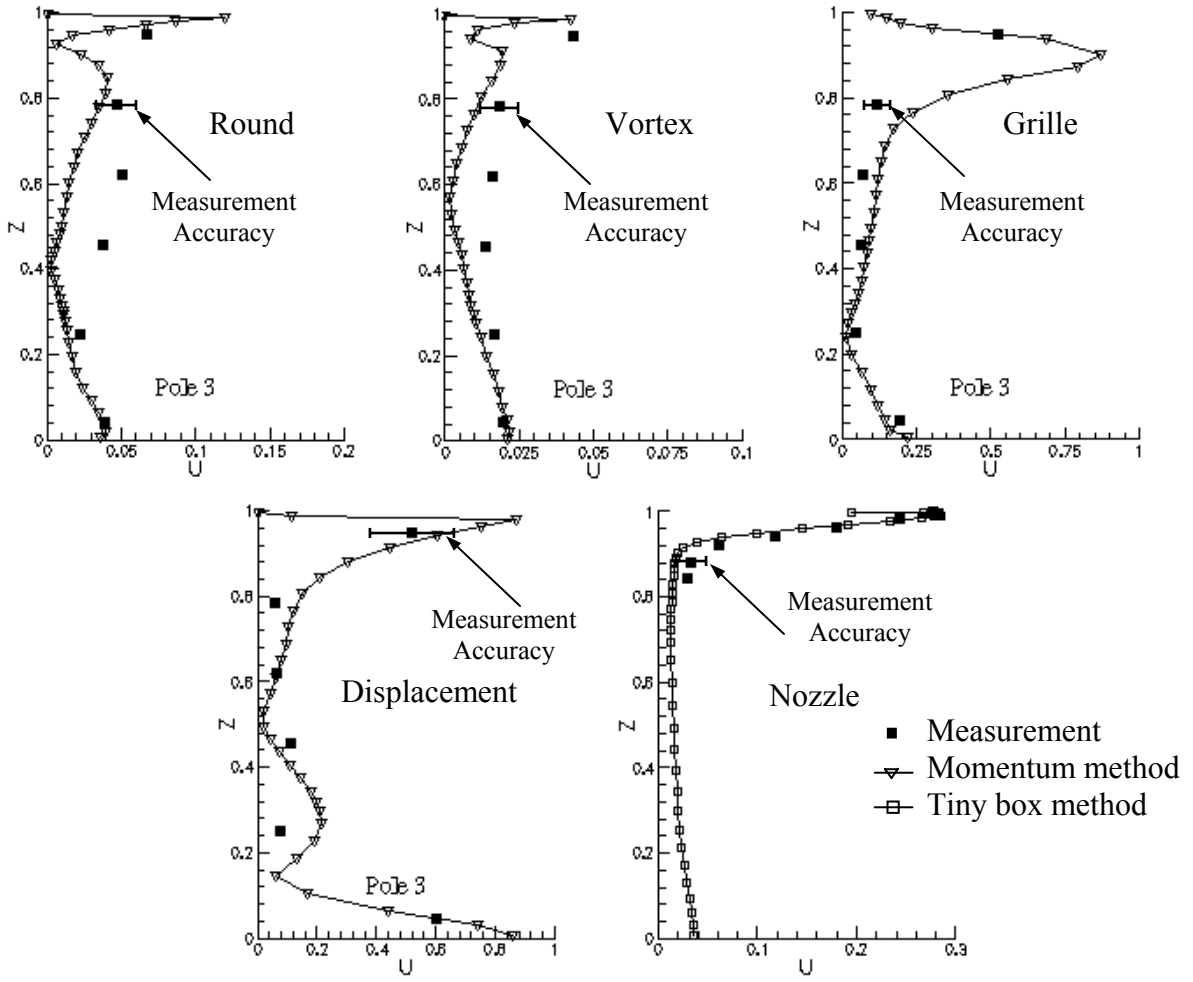


Figure 13. The comparison of the calculated and measured velocity profiles for the round, vortex, grille, displacement, and nozzle in a typical location in a room. Z =height/total room height (H), U =velocity/supply velocity (U_0).