Srebric, J. and Chen, Q. 2001. “A method of test to obtain diffuser data for CFD modeling of room airflow,” *ASHRAE Transactions*, 107(2), 108-116.

A Method of Test to Obtain Diffuser Data for CFD Modeling of Room Airflow (RP-1009)

Jelena Srebric, Ph.D.
Student Member ASHRAE

Qingyan Chen, Ph.D.
Member ASHRAE

ABSTRACT

This study demonstrated how to use two simplified methods, the box and momentum methods, to simulate complex diffusers in room airflow modeling by computational fluid dynamics. The two methods require additional flow information that is not available from product catalogues of diffuser manufacturers. This information includes the distribution of discharge air velocity, box size, and flow direction. This research developed a method of test (MOT) on how to obtain the additional information with minimal effort under the ASHRAE Standard 70-1991.

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

---

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

INTRODUCTION

When using computational fluid dynamics (CFD) for room airflow design, there are several factors that constrain the CFD applications. The reliability and accuracy of CFD simulations depends on several factors, such as the turbulence model, the numerical scheme, and the boundary condition modeling. Correct modeling of the supply air boundary condition is most crucial, because the momentum from the supply air diffuser mainly dominates the room airflow. Since a diffuser is much smaller than a room, to model the same detail for the diffuser and the room would require a very different length scale in the CFD modeling. Since the diffuser determines the initial jet flow characteristic, several researchers (Emvin and Davidson 1996, Chen and Jiang 1996) have chosen to simulate detailed diffuser geometry. To model detailed diffuser geometry would require millions of grid cells, which in turn require a large computer capacity, but this does not guarantee a successful simulation. On the other hand, to use the room length scale would ignore the details in the diffuser, which could introduce errors in the numerical simulation. Therefore, an alternative solution is to develop a simplified but reliable method to simulate a diffuser while allowing the use of a room-length scale in CFD simulations.

With numerous studies conducted elsewhere (IEA 1993, Emvin and Davison 1996, Chen and Jiang 1996, Nielsen 1997, and Huo et al. 1996), Chen and Srebric (2000) have investigated systematically several simplified modeling methods for complex air diffusers. They have identified two simplified methods, the box and momentum methods, to be most appropriate for use in CFD simulations of indoor airflow. When the box method is used in a CFD simulation, it needs the distributions of air velocity, air temperature, and contaminant concentrations around the diffuser. The box size should also be correctly determined. Similarly, the momentum method requires the airflow rate, discharge jet velocity or effective diffuser area, supply air turbulence properties, supply air temperature, and contaminant concentrations. Unfortunately, not all the parameters are available from the product catalogues of the diffuser manufacturers, nor can they be easily obtained from calculations. It is necessary to obtain some of these data through experimental measurements. The objective of this paper is to present a method of test (MOT) for obtaining the data.

SIMULATION OF A DIFFUSER WITH SIMPLIFIED METHODS

This section will demonstrate how to use the box and momentum methods to simulate a complex nozzle diffuser for room airflow modeling, as an example to show the needs for obtaining the data. The study uses a commercial CFD program (CHAM 1998). The demonstration will show how to apply the box and momentum methods and what data are needed in order to develop a suitable MOT.

The nozzle diffuser is the one used for validation exercises in an international project (IEA 1993). The diffuser consists of 84 small round nozzles arranged in four rows in an area of 28 in. × 6.7 in. (0.71 m × 0.17 m), as shown in Figure 1(a). Each nozzle has a diameter of ½ in. (11.8 mm), and the total airflow area for all nozzles is 0.1 ft^2 (0.0092 m^2). The flow direction of each nozzle is adjustable. The IEA study adjusted all the nozzles 40^o upwards. The diffuser was installed close to the ceiling in the center of the shorter wall in an empty test room, which is 15.8 ft (4.8 m) long, 9.8 ft (3.0 m) wide, and 8.2 ft (2.5 m) high, as shown in Figure 1(b). An exhaust was installed underneath the nozzle diffuser.

Ewert et al. (1991) measured the air velocity using a laser Doppler anemometer (LDA). Since the LDA measured the velocity components in three directions, the flow directions are known. The supply airflow rate was 67 cfm (0.0315 m^3/s), corresponding to an air change rate of 3 ACH and a maximum jet velocity of 760 fpm (3.8 m/s) at the supply plane. The decay of this supply velocity was very fast due to the merge of the 84 small jets. The size of the recirculation zone was approximately 8 in. x 8 in. (0.2 x 0.2 m), as shown in Figure 2. The maximum jet velocity dropped to 300 fpm (1.5 m/s) at approximately 4 in. (0.1 m) from the diffuser, where the small jets were already combined into one large jet. The momentum loss measured in front of the diffuser was approximately 14% of the total momentum (Heikkinen 1991). Hence, the momentum was not conserved. All the experimental data has been used in the present study to validate the numerical results.

Box Method

When the box method is used to simulate the diffuser for room airflow modeling, it is necessary to specify the distributions of air velocity, air temperature, and contaminant concentrations in the box surface through which the flow is discharged. The best approach is to find a suitable jet formula to calculate the distributions. For isothermal attached jets, Verhoff (1963) proposed the following formula that is extensively used for diffuser jets (Skovgaard et al. 1990, Jacobsen and Nielsen 1992):

$$\frac{u}{u_m} = 1.48\eta^{1/7}\left[1 - \mathrm{erf}(0.68\eta)\right] \qquad (1)$$

where u = the jet velocity in direction x at a distance y from the wall

u_m = the maximum jet velocity in the x direction

$$\eta = \frac{y}{y_{1/2Um}}$$

$y_{1/2Um} = 0.073(x + 12b)$ (Rodi 1982), $y_{1/2Um} = 0.08(x + 0.45)$ (Skovgaard et al. 1990)

x = the distance from the jet origin

b = diffuser coefficient

Although the jet from the diffuser attached to the ceiling at approximately x = 8 in. (0.2 m), it did not immediately develop into a normal attached jet. At 1.0 m (3.3 ft) from the diffuser, the jet flowed parallel to the ceiling, but the jet profiles cannot be described by Equation (1) as an attached jet. The jet formula can only be applied at x = 7.2 ft (2.2 m), where the jet profile can be well predicted, as shown in Figure 3. However, the distance is too large for the box method because the box would cover half of the ceiling length, and would influence the numerical solution of elliptic equations (Emvin and Davidson1996). Under a non-isothermal condition, the thermal plumes in the room would have a strong interaction with the jet, and the formula also cannot be applied. Therefore, the jet formula is not very useful for most practical HVAC applications.

Since the jet formula cannot be used, the box method needs measured data to set the boundary conditions. The question is at which location the flow data should be measured. In other words, how big should the box be? In previous studies, such as the one conducted by IEA (1993), the box size was selected to be sufficiently small to avoid the impact of the room size and thermal plumes. However, the box size could be larger than the domain with the recirculation. Then, the calculation does not need to handle the difficult estimation of flow in this recirculation area.

The commonly used box size, such as the one used by Heikkinen (1991) and Ewert et al. (1991), is $\Delta x \times \Delta y \times \Delta z$ = 40 in. × 20 in. × 14 in. (1.0 m × 0.5 m × 0.4 m), as shown in Figure 4(a). With this box size, the velocity profile in the front surface where flow was discharged should be described as the boundary conditions for the CFD simulation. The boundary conditions for all the variables at all other surfaces can be assumed to be zero-gradient. The measured velocity, as shown in Figure 5, was used to set the boundary conditions for the box method. The data shows that the jet had a very strong three-dimensional feature. Our simulations used 3 × 3 supply-patches, as illustrated in Figure 4(a). Each of supply-patch used a velocity averaged from the measured data for the represented area.

The figure on the right in Figure 6 compares the calculated results with the measured data in the middle section of the room at 7.2 ft (2.2 m) from the diffuser. In the upper part of the room, the CFD over-predicted the jet velocity by 25%. The reason for the discrepancies might be that the box was too large, and the 3× 3 supply patches may be too coarse for such a large box. With such a large box, it is impossible for us to validate the calculated results at 3.3 ft (1.0 m) from the diffuser, where the experimental data is available. At this location, the box size is the same as the distance.

Therefore, the next simulation used a smaller box that neglects the lower part of the measured velocity profiles, as shown in Figure 4(b). The justification for this approximation is that the jet in the upper part of the box is responsible for most of the momentum transport. Although the results are not presented here, the smaller box does not improve the results much.

Finally, this study used a tiny box 12 in. x 16 in. x 10 in. (0.3 m x 0.4 m x 0.25 m), as shown in Figure 4(c). The box is so small that one has to impose a velocity profile at the top surface of the box as the boundary conditions for the CFD simulation. This is quite different from the traditional approaches where box is located in the fully-developed jet region. The jet in the tiny box surface may not be fully developed. In fact, the velocity profile is only imposed on the front part of the top surface, where the flow is outwards. At the front part of the top surface, the flow direction is 45^{o} upwards with a uniform air supply velocity (one patch). In this case, it just happens that the velocity is uniform at the surface. Note that this is not a general phenomenon. The supply velocity and flow direction used for the tiny box was estimated from the data and smoke visualization from Hekkinen (1991), as shown in Figure 2.

The airflow pattern calculated with the tiny box method, as shown in Figure 7(a), has a recirculation in the upper left corner. The recirculation is approximately the same size as the one observed by the smoke visualization. This is the only method that is able to predict correctly airflow patterns close to the diffuser. Figure 8 further compares the computed velocity profiles with the measured ones in two sections at the middle plane of the room. The agreement between the computed and measured results is excellent. Therefore, the tiny box method is appropriate for a nozzle diffuser simulation, since the airflow pattern and jet profiles are correctly simulated.

The study demonstrates that the box size is crucial for correct prediction of the room airflow with the nozzle diffuser. The box size is also important for any other diffuser simulated by this method. The second part of the paper will present a method of test to determine the box size. In addition, the example shows it is necessary to provide the air velocity and flow direction at the box surface. Furthermore, the air temperature and contaminant concentrations must be provided for non-isothermal conditions and for situations with species transport. How to obtain these data will be also the subject of the second part of the paper.

Momentum Method

Another simplified method used to simulate the nozzle diffuser is the momentum method. It is very simple to simulate the diffuser with the momentum method in a CFD program. Our study, for example, specifies at the diffuser area a supply airflow rate of 67 cfm (0.0315 m^3/s), and a discharge velocity of 760 fpm (3.8 m/s), directed 40^{o} towards the ceiling (see Figure 4(d)).

Figure 7(b) shows no recirculation computed by this method in the upper left corner, and the jet spreads very rapidly in the region after the attachment to the ceiling. This contrasts the experimental observation shown in Figure 2. Because of this problem, this investigation has further calculated with finer grid resolutions, different turbulence models, and different numerical schemes (Srebric 2000 and Chen and Srebric 2000). Unfortunately, all of these attempts failed to predict the recirculation.

Nevertheless, Figure 6 shows that the momentum method performed reasonably in predicting the velocity profiles at the two sections, although the method failed to predict the recirculation in the upper-left corner of the room. The momentum method is better than the commonly used large box method.

Our results (Chen and Srebric 2000) have also concluded that the momentum method can successfully be used to simulate the following five types of diffusers: displacement, square ceiling, round ceiling, vortex ceiling, and grille diffusers. Therefore, it is important to know how to use the momentum method, and it is discussed in the following section.

In order to apply correctly the momentum method, crucial parameters needed are the discharge velocity from the diffusers, U_o, or the diffuser effective area, A_o, as well as the discharge flow direction. The U_o and A_o are interrelated as

$$U_o = \frac{\dot{m}}{\rho A_o} \qquad (2)$$

where $\dot{m}$ = mass flow rate from the diffuser

ρ = air density

The next section will present a method to determine the U_o or the A_o, and the flow direction.

METHOD OF TEST

Our experience with the simulations of the eight complex diffusers (Srebric 2000 and Chen and Srebric 2000) found that the box and momentum methods could be used to simulate complex diffusers. With the methods, there is no need to specify the detailed diffuser geometry in a CFD simulation of room airflow. The data used in the methods are the diffuser geometry, which is normally available from the product catalogues of the diffuser manufacturers, and the flow field in the diffuser vicinity. More specifically, the box method needs the distribution of the jet velocity, flow direction, air temperature, and contaminant concentrations on the box surface, and the momentum method requires the discharge velocity and flow direction. Unfortunately, these parameters on the box surface are not available in the current product catalogues; they need to be obtained through measurements. This section describes a method of test (MOT) on how to obtain these parameters.

Box Method

The original box method requires measured data for all the simulated variables in order to set the diffuser boundary conditions. If a simulation is isothermal, only the velocity distribution and flow direction needs to be measured for the box method. However, if the simulated space needs information on temperature and contaminant concentration distributions, the temperature and concentration profiles also need to be measured on the box surface. This is not a trivial job. Figure 9 shows the tracer-gas distribution measured at two locations in front of a grille diffuser installed in an office (Srebric 2000 and Chen and Srebric 2000). The concentration measurements required a significant labor effort and expensive equipment.

Our study (Srebric 2000 and Chen and Srebric 2000) found that the resulting distributions of air temperature and tracer-gas concentration in a room are not very sensitive to the profiles of the temperature and tracer-gas concentrations at the box surface. The use of the average temperature and concentrations is an acceptable and practical approach to impose boundary conditions for the box method. This implies that only the velocity profile needs to be specified in the box method, and the other parameters can be set as uniform. Note that the velocity distribution at the box surface cannot normally be assumed to be uniform, otherwise the box method cannot correctly predict air parameters in the jet region. The uniform air temperature T_{box} and tracer gas concentrations C_{box} can be determined from the mass, energy, and species concentration balances at the box boundaries (see Figure 10 for more information):

$$\Delta\dot{m} = \dot{m}_{box} - \dot{m}_{supply} \quad (3)$$

$$T_{box} = (\dot{m}_{supply} T_{supply} + \Delta\dot{m}\, T_{exhaust}) / \dot{m}_{box} \quad (4)$$

$$C_{box} = (\dot{m}_{supply} C_{supply} + \Delta\dot{m}\, C_{exhaust}) / \dot{m}_{box} \quad (5)$$

where $\dot{m}$ is flow rate and $T_{exhaust}$ and $C_{exhaust}$ are the temperature and contaminant concentration at the room exhaust, respectively.

With the velocity distribution and the uniform temperature and tracer gas concentration at the box surface, the temperature and tracer gas concentration distributions in the jet region can be correctly predicted. We (Chen and Srebric 2000) have documented some examples.

Since the boundary conditions for the air temperature and the tracer gas concentration can be specified as uniform, the measurements needed to obtain the velocity profile can be performed under isothermal conditions. The velocity distribution needs to be measured at quite a few points (at least six to fifteen) in the jet discharge direction. The measurements should be performed with low-velocity anemometers, as described in the ASHRAE Standard 70-1991 "Method of Testing for Rating Performance of Air Outlets and Intakes". The standard gives an MOT for rating the performance of air outlets and inlets based on the jet throw. The MOT specified here complies with ASHRAE requirements for velocity measurements. In addition, smoke visualization is needed to determine jet flow direction at the box surface.

The measurements with the hot sphere/wire anemometers and smoke visualization are sufficient to impose reliable boundary conditions for a diffuser with the box method. Measurements with more sophisticated equipment such as LDA are expensive. The equipment is not a standard measurement technique in diffuser manufacturers' laboratories. In fact, our calculations for the nozzle diffuser indicate that the hot-wire anemometer measurements and smoke visualization provide a more important source of information than the data obtained with the LDA.

In addition to the distributions of air velocity, flow direction, air temperature, and contaminant concentrations, the box size is another important parameter, as discussed in the previous section.

If the diffuser discharges multiple jets, such as the nozzle diffuser, the box size should be in the region where the diffuser jets have merged. The CFD with Reynolds averaged Navier-Stokes equations has difficulty simulating flow in the region before the jets have merged. The minimal box size can be roughly estimated from smoke visualization. In the example shown in the previous section, the tiny box size is 16 in. (0.4 m) in the x-direction. Since the jets merged at x = 4 in. (0.1 m), the tiny box method satisfied the minimum requirement. On the other hand, the box size should not be too large, since a large box may cover a portion of the domain that is important for thermal plumes and obstacles in the room, due to a strong buoyancy and contained flow effect.

Although the box data gathered under isothermal conditions can be applicable for non-isothermal conditions, the box size should be selected in such a way that the buoyancy force is negligible compared to the momentum force at the box surface. A practical criterion to compare these forces is the local Archimedes number, Ar_x, defined by Grimitlyn and Pozin (1993). If the local Archimedes number is smaller than a certain value, the jet is weakly non-isothermal and can be modeled by the isothermal jet formulae (Srebric 2000 and Chen and Srebric 2000). From the work of Grimitlyn and Pozin (1993), we have derived a critical distance x_c, where the jet velocities are still predominately influenced by the momentum forces. Within that distance, the jet velocities obtained from the isothermal experiment can be used as the boundary conditions in the non-isothermal simulations.

The box size should be smaller than x_c from the jet origin in the discharge direction:

☞ Plane (linear) jets

$$x_c \le H_o \left(\frac{K_1^2}{K_2} \frac{0.15}{Ar_o} \right)^{2/3} \tag{6}$$

☞ Axisymmetric and radial jets

$$x_c \le K_1 \left(\frac{0.1}{K_2} \frac{A_o}{Ar_o} \right)^{1/2} \tag{7}$$

where K_1 is the centerline velocity decay constant, K_2 is the centerline temperature decay constant, and Ar_o is the Archimedes number based on the air supply parameters. The centerline velocity and temperature decay constants are available in the literature for typical diffuser jet flows, such as ASHRAE (1997) and Li et al. (1993).

Note that x_c is not the box size recommend. The x_c is the maximum box size allowed. The actual box size should be at the location where the jets have merged. If it is a single jet diffuser, the box should be as close to the diffuser as possible.

The criterion in Equations 6 and 7 implies that the box method is not suitable for low Reynolds number flows, such as displacement ventilation. For displacement ventilation, the buoyancy force strongly influences the jet development from its discharge. The jet from the displacement diffuser changes its profile shape and position very rapidly in front of the diffuser (Jacobsen and Nielsen 1992). On the other hand, in the case of mixing ventilation, the criterion should be easy to satisfy with a relatively small box.

For example, the grille diffuser (Figure 9(a)) has K_1=5.7 and K_2=3.4. The calculated x_c is 3.3 ft (1.0 m) for 5 ACH, and 1.8 ft (0.56 m) for 3 ACH. A slot diffuser (K_1=2.2, K_2=1.7) has x_c = 7.3 m (Chen and Srebric 2000). The centerline velocity and temperature constants (K_1 and K_2) are not available for the studied nozzle diffuser. However, the nozzle diffuser is with isothermal flow (Ar_o=0), and therefore the criterion for x_c is always satisfied. For most mixing ventilation diffusers, the x_c is very large because of its relatively high discharge jet velocity. Therefore, the buoyancy force can be safely neglected in the diffuser vicinity.

Based on the above analysis, the MOT for obtaining boundary conditions for the box method can be summarized as follows:

- ☞ The room size should comply with the ASHRAE Standard 70-1991 for the throw test
- ☞ The required equipment are low velocity anemometers and smoke visualization devices
- ☞ The test can be performed under isothermal conditions
- ☞ Air velocities should be measured at 6 to15 points in the jet region at the box surface where the jet impinges, as shown in Figure 10. Fewer measuring points are used for jets with a uniform velocity distribution at the box surface, and more points are used for a non-uniform velocity distribution.
- ☞ The box surface should be in the region where the jets have merged, as close as possible to the diffuser

The data from the MOT and the following procedure are necessary to set boundary conditions for a diffuser in CFD simulations:

- ☞ Check if the jet is weakly non-isothermal in the box surface (with Equations (6) and (7) or perform one additional measurement for non-isothermal conditions)
- ☞ Use the balance equations to determine T_{box} and C_{box} (Equations (3), (4) and (5))
- ☞ Define box patches with the measured velocities and set other supply air parameters

The box method defined here is a robust modeling method. The momentum from the induced jet flow on the other surfaces of the box boundary can be safely neglected. The reason is that the high jet momentum on the box surface normal to the jet is mainly responsible for most of the mass, energy, and species transport. Also, the averaged velocity for each patch is sufficient as boundary conditions for the diffuser. Therefore, there is no need to measure in great detail the velocity profiles on the box surface. The box method is better for using with the mixing diffusers, with which the local Archimedes number is low. However, the method is not suitable for the displacement diffuser because of the high Archimedes number.

With the recommended procedures above, one would be able to obtain the distribution of the discharge velocity and flow direction at a box surface, the suitable size of the box, and the mean air temperature and contaminant concentrations. These data can be used as boundary conditions from a diffuser for a CFD simulation of room airflow.

Momentum Method

The ASHRAE Standard 70-1991 specifies a procedure, a test chamber, and instruments for the measurements of the diffuser jet throw, pressure drop, and discharge velocity. From the measurements of the discharge velocity and flow rate, manufacturers calculate an effective diffuser area called the area factor. In fact, the area factor is a function of the airflow rate. If the area factor changes more than 5% with the flow rate, the manufacturers should report the change. Only a few manufacturers provide information on the area factor, but not the measuring positions. Therefore, it is difficult to use the area factor to determine the discharge velocity for use in the momentum method.

The measurements for the nozzle diffuser (Skovgaard et al. 1990) found that the area factor depended on the Reynolds number, i.e., the flow rate. This further complicates the problem. In order to confirm that the area factor is a function of the flow rate, we have further tested a grille diffuser (see Figure 9(a)) in an environmental chamber of 17 ft (5.16 m) long, 12 ft (3.65 m) wide, and 8 ft (2.43 m) high. The nozzle diffuser and the grille diffuser represent the two extremes of the geometry complexity among the eight diffusers studied (Srebric 2000 and Chen and Srebric 2000) - the nozzle is the most complex and the grille is the simplest.

The flow rate tested for the grille diffuser was 3, 5, and 8 ACH, respectively. This study measured discharge jet velocities in front of the diffuser, and found that the effective area changes from 4% to 5% with the three flow rates, as shown in Table 1. Note that the area factor measured A_o is not the same as that calculated from the geometry of the diffusers A_{og}. For the nozzle diffuser, the difference between the two is about 10%. For a slot diffuser, it can be more than 100%. Therefore, the effective area ratio, ε, is not a constant. This effective area ratio is a function of the flow rate.

Cheong et al. (1999) studied a square diffuser and found that the results are better if the calculations use the measured discharge velocity rather than the calculated discharge velocity from the diffuser geometry. Their study confirms once again that the area factor cannot be correctly estimated from the diffuser geometry. Therefore, it is very important for diffuser manufacturers to measure the area factor and to report the corresponding measuring positions along with other parameters described in the ASHRAE Standard 70-1991.

Furthermore, the measured data for the grille diffuser shows that the discharge velocity is not uniform (see Figure 11). The jet had a higher momentum in the lower left corner of the diffuser. Since it is the intention of this investigation to use realistic ducting in the study, the straight duct behind the diffuser is less than 10 times the equivalent of the diffuser diameter. As a result, the non-uniformity of the velocity distribution is approximately $\pm$ 18% for the three flow rates studied. This non-uniformity has also been observed in other diffusers and test facilities (Srebric 2000 and Chen and Srebric 2000).

In order to measure a mean discharge velocity (or area factor) for a diffuser, Figure 12 recommends where the discharge velocity should be measured. Figure 12(a) is for a displacement diffuser and the grille diffusers, and Figure 12(b) is for ceiling types of diffusers, such as square, round, and vortex diffusers. Measurements at multiple points are suggested in actual room installations because of the possible asymmetry on the flow distribution. In manufacturers' laboratories, the measuring points can be reduced if their system can produce a uniform distribution. In addition to the installation of the diffuser, different manufacturing processes can have an influence on the effective area factor, as reported by Nielsen (1991) and

Knappmiller (1994). A careful adjustment after installation can help avoid the asymmetry of the flow discharge.

Finally, the momentum method needs to know the flow direction discharged from a diffuser. This can be easily done by smoke visualization.

In summary, the MOT for obtaining boundary conditions for the momentum method uses the same equipment as the MOT for the box method, and the procedure is:

- Measure discharge velocity in 8 or 9 points close to the diffuser as shown in Figure 12
- Determine the discharge flow direction from smoke visualization

By following the above-recommended procedures, a correct discharge velocity or effective area of a diffuser and flow direction can be obtained as boundary conditions from the diffuser in a CFD simulation of room airflow.

CONCLUSIONS

By computing room airflow distributions with a complex nozzle diffuser using CFD, this study demonstrated how to use the simplified box and momentum methods to simulate the diffuser. The results show that it is possible to use the simplified methods to obtain correct indoor air distribution.

However, the simplified box and momentum methods require measurements of some parameters necessary for the computations. A method of test (MOT) has been developed in order to provide the experimental data. The following are conclusions obtained from this study:

- The measured velocities at the box surface are used to define mass flow and to calculate energy and species balance for the box method. The measured velocities under isothermal conditions can be used for non-isothermal flow, if the local Ar number is sufficiently small. A uniform profile of air temperature and contaminant concentration is acceptable for setting diffuser boundary conditions.
- The study recommends a minimum box size, which should be located in the region where the diffuser jets have merged. On the other hand, the box size should not be too large in order to avoid the impact of thermal plumes, room size, and internal obstacles on the jet development. An equation has been developed to estimate the maximum box size.
- The momentum method requests the discharge jet velocity and flow direction as boundary conditions for simulating a diffuser. The discharge velocity can be a function of the flow rate and can be asymmetric on the diffuser surface. This investigation recommends a procedure on how to determine the discharge velocity. The flow direction can be estimated by smoke visualization.
- The ASHRAE Standard 70-1991 describes a procedure for testing air supply diffusers. This study does not require a more substantial effort than that defined in the ASHRAE standard, but a more precise definition on where and how to measure. Therefore, diffuser manufacturers can easily implement the MOT.

REFERENCES

ASHRAE. 1991. Method of testing for rating the performance of air outlets and inlets, ANSI/ASHRAE Standard 70 - 1991, Atlanta: ASHRAE.

ASHRAE. 1997. ASHRAE Handbook - Fundamentals, ASHRAE, Atlanta, Ch.31.

CHAM. 1998. PHOENICS Version 3.1, CHAM Ltd., London, UK.

Chen, Q., and Jiang, Z. 1996. Simulation of a complex air diffuser with CFD technique, Proc. of ROOMVENT '96, Vol. 1, pp. 227-234.

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.

Cheong, K.W.D., Sekhar, S.C., Tham, K.W., and Djunaedy, E. 1999. Airflow pattern in an air-conditioned seminar room, Indoor Air ’99, Vol.2: 54-59.

Emvin, P. and Davidson L. 1996. A numerical comparison of the diffuse in case E1 Annex20, Proc. of ROOMVENT '96, Vol. 1: 219-226.

Ewert, M., Renz, U., Vogl, N., and Zeller, M. 1991. Definition of the flow parameters at the room inlet devices - measurements and calculations, Proc. of 12th AIVC Conference, Vol. 3: 231-237.

Grimitlyn, M.I. and Pozin G.M. 1993. Fundamentals of optimizing air distribution in ventilated spaces, ASHRAE Trans., 99(1): 1128-1138.

Heikkinen, J. 1991. Modeling of a supply air terminal for room air flow simulation, Proc. of 12th AIVC Conference, Vol. 3: 213-230.

Huo, Y., Zhang, J., Shaw, C., and Haghighat, 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. 1992. Velocity and temperature distribution in flow from an inlet device in rooms with displacement ventilation, Proc. of ROOMVENT '92.

Knappmiller, K.D. 1994. The mixing and separation characteristics of cold air ceiling jets, Ph.D. Dissertation, Colorado State University, Fort Collins, Colorado.

Li, Z.H., Zhang, J.S., Zhivov, A.M., and Christianson, L.L. 1993. Characteristics of diffuser air jets and airflow in the occupied regions of mechanically ventilated rooms - a literature review, ASHRAE Trans., 99(1): 1119-1127.

Nielsen, P.V. 1991. Models for the prediction of room air distribution, Proc. of 12th AIVC Conference, Vol. 1: 55-71.

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.

Rodi, W. 1982. Turbulent buoyant jets and plumes, Pergamon Press, Oxford, 1982.

Skovgaard, M., Hyldgaard, P. and Nielsen, P.V. 1990. High and low Reynolds number measurements in a room with an impinging isothermal jet, Proc. of ROOMVENT '90.

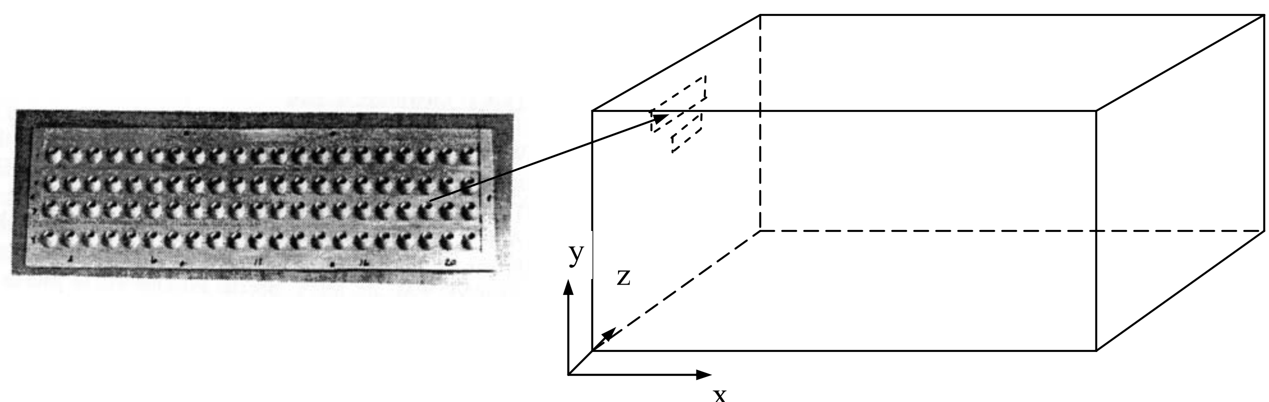
Srebric, J. 2000. Simplified methodology for indoor environment design, Ph.D. Thesis, Department of Architecture, Massachusetts Institute of Technology, MA.

Verhoff, A. 1963. The two-dimensional turbulent wall jet with and without an external stream, Report 626, Princeton University.

Table 1. The measured and calculated diffuser geometry and flow parameters

Diffusers	Air exchange Rate ACH (kg/s)	Supply velocity fpm (m/s)	Gross area (A_{gross}) m^2	Area factor (A_o) * m^2	$\varepsilon=A_o/A_{gross}$ %	Geom. Area Factor (A_{og})** m^2
Grille	8.0 (0.1228)	429 (2.18)	0.28x0.18	0.0469	93	0.0336
	5.0 (0.0768)	276 (1.40)	0.28x0.18	0.0457	91	0.0336
	3.0 (0.0456)	169 (0.86)	0.28x0.18	0.0442	88	0.0336
Nozzle	3.0 (0.0378)	748 (3.80)	0.71x0.17	0.0083	7	0.0092

* The measured area factor
** The area factor calculated from the diffuser geometry



(a) (b)

Figure 1. Sketch of the nozzle diffuser and the test room.

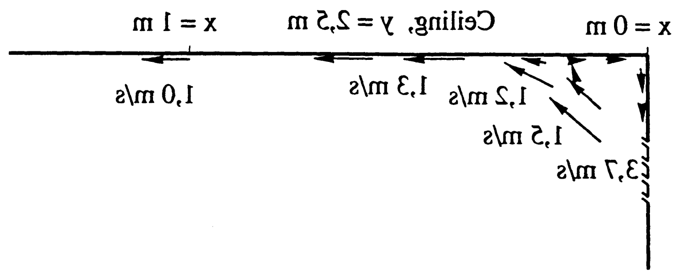


Figure 2. Airflow pattern visualized with smoke near the diffuser in the symmetry plane (Heikkinen 1991).

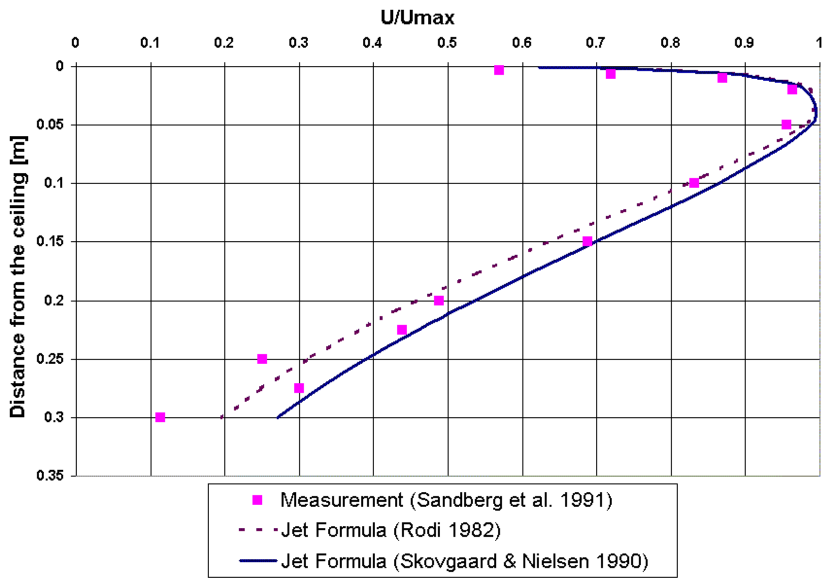


Figure 3. Attached jet developed closed to the ceiling at 7.2 ft (2.2 m) from the nozzle diffuser.

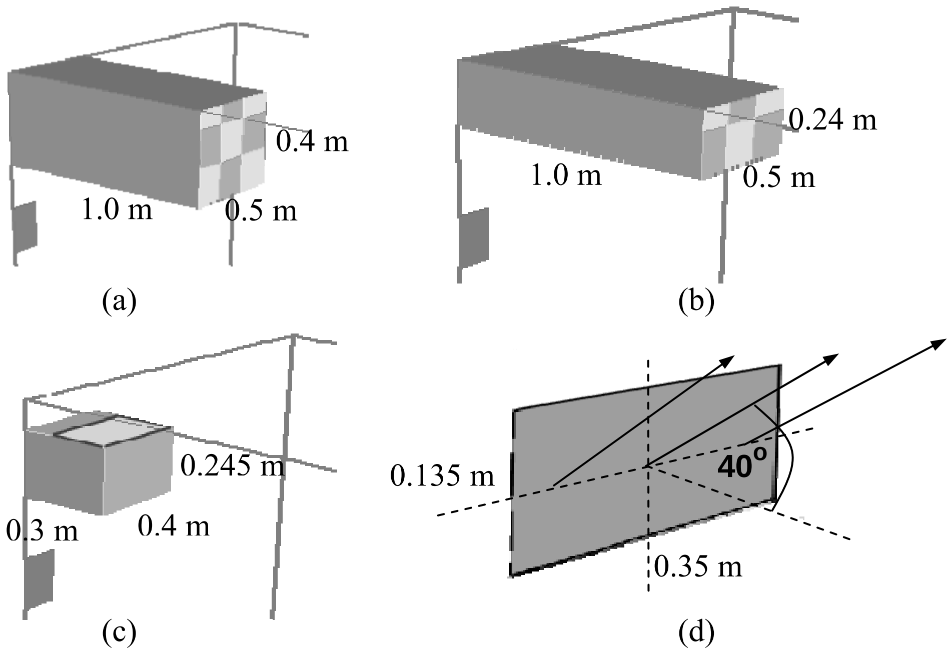


Figure 4. Simulation of the nozzle diffuser with (a) the commonly used box method (b) the small box method, (c) the tiny box method, and (d) the momentum method

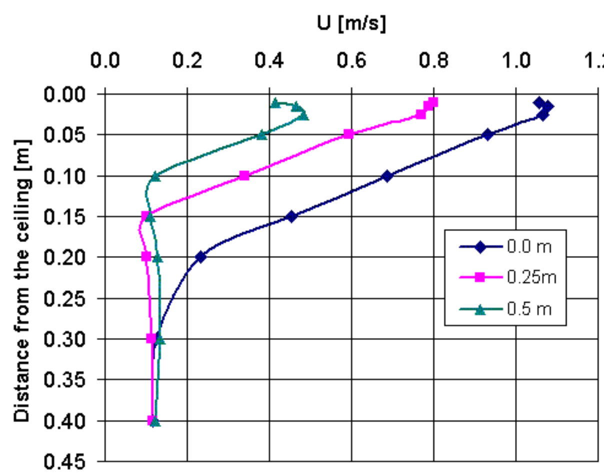


Figure 5. The measured air velocities at different distances from the middle plane of the room at x = 1.0 m (3.3 ft) in front of the diffuser and

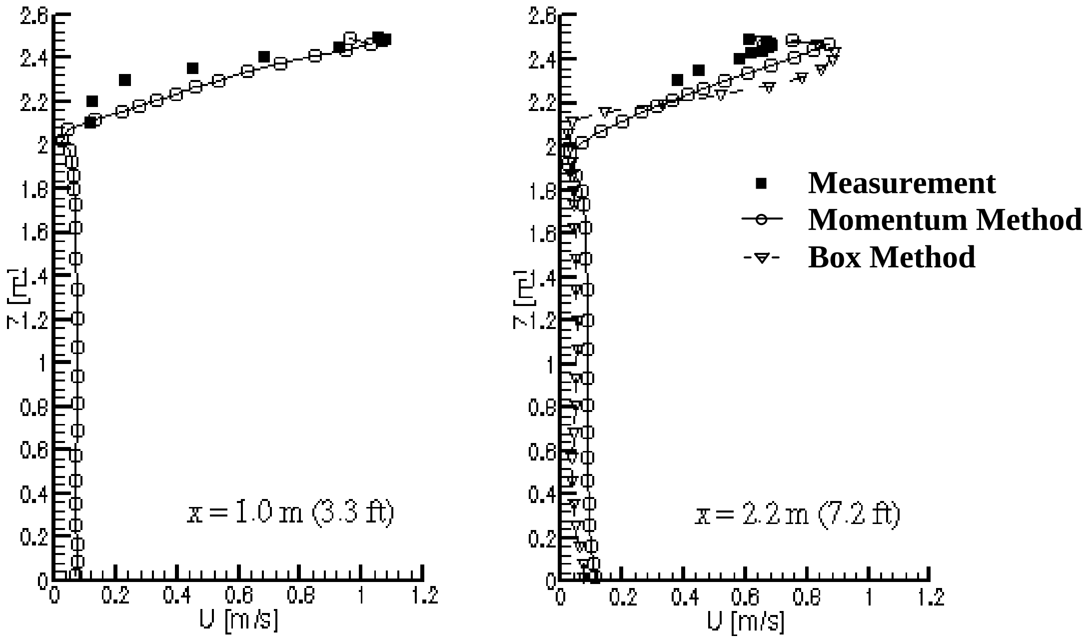


Figure 6. The comparison of the calculated and measured velocity profiles for the nozzle diffuser at the two positions in the middle plane of the room.

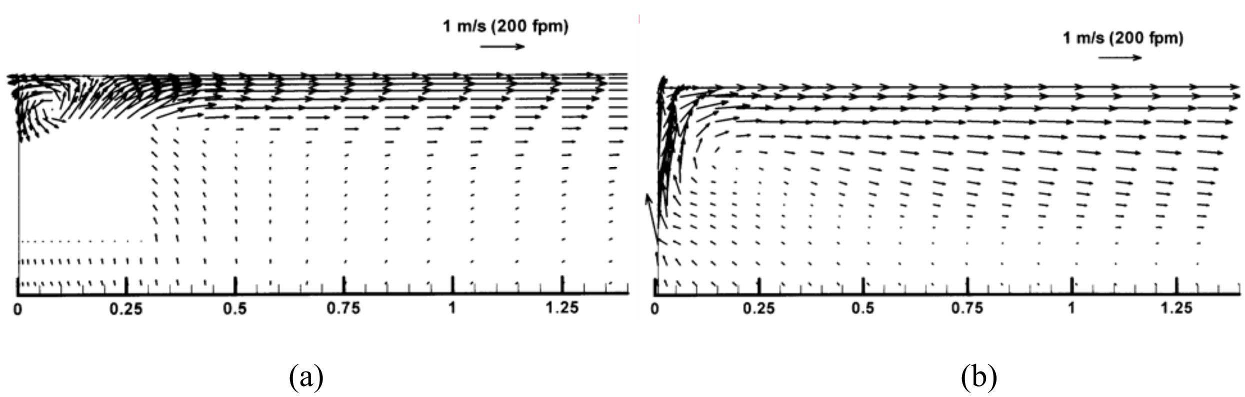


Figure 7. The calculated flow field with (a) the tiny box method and (b) the momentum method.

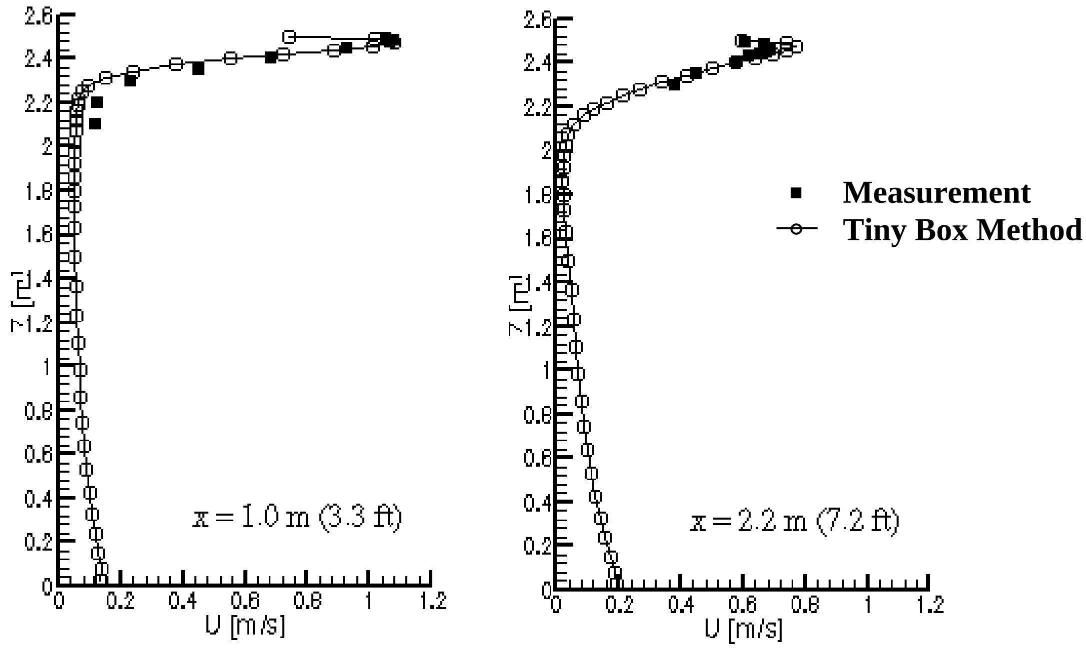
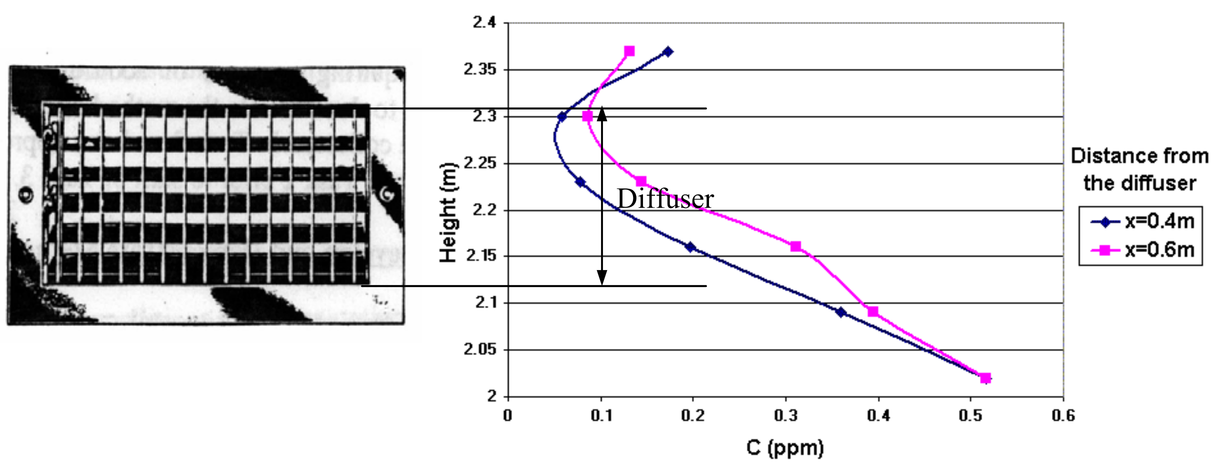
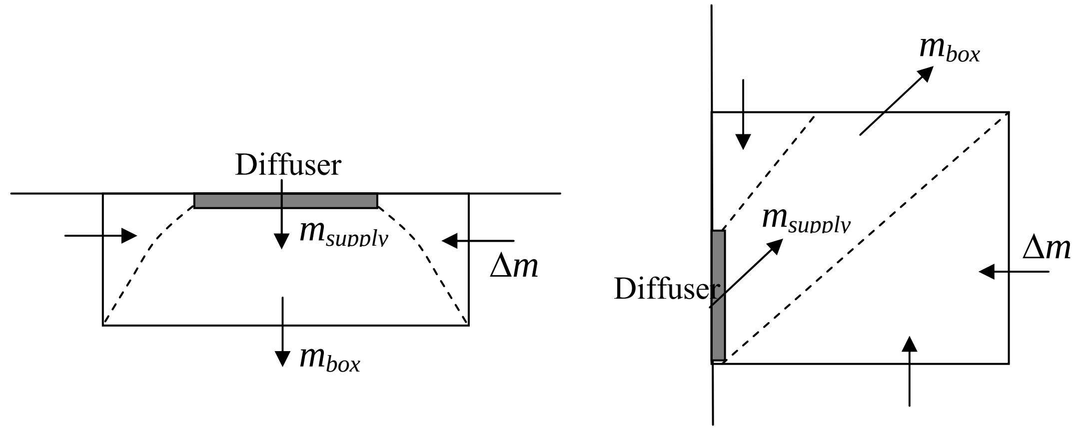


Figure 8. The comparison of the calculated and measured velocity profiles for the nozzle diffuser at two positions in the middle plane of the room.



(a) (b)

Figure 9. A typical tracer-gas distribution at two sections in front of a grille diffuser



(a) (b)

Figure 10. Sketch of the mass balance at the box boundaries for (a) a downward free jet and (b) an inclined jet.

0.70 •	**0.65** •	**0.72** •
0.98 •	**0.90** •	**0.88** •
1.00 •	**0.96** •	**0.90** •

3 ACH

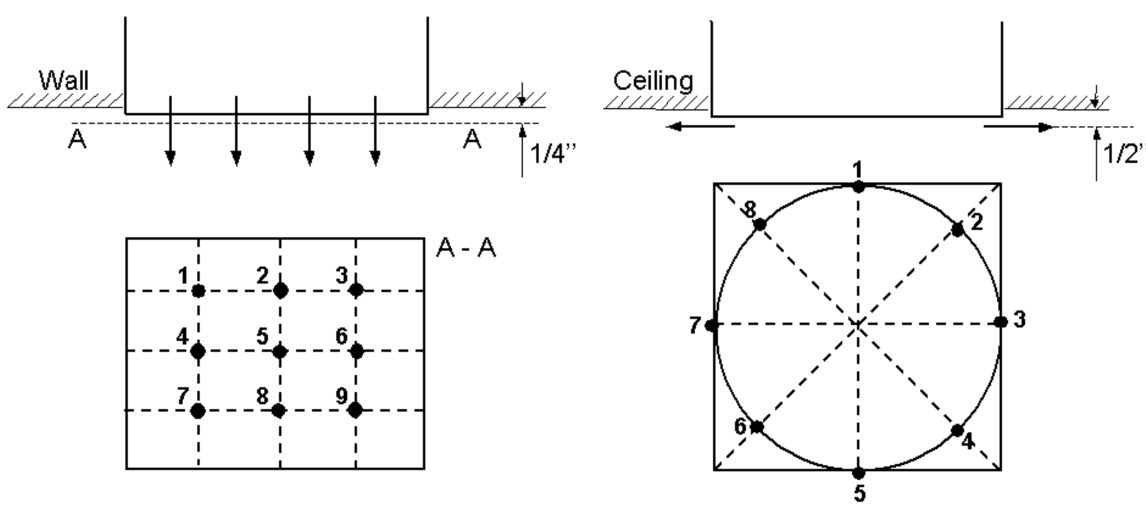
1.20 •	**1.13** •	**1.30** •
1.50 •	**1.45** •	**1.38** •
1.65 •	**1.52** •	**1.43** •

5 ACH

1.90 •	**1.65** •	**2.05** •
2.44 •	**2.20** •	**2.15** •
2.50 •	**2.40** •	**2.28** •

8 ACH

Figure 11. The measured discharged jet velocities in front of the grille diffuser for different airflow rates (m/s)



(a) (b)

Figure 12 Recommended positions for measurements of the discharge velocity.