Examination of the Ventilation Function of a Combined Air-Diffuser Ventilation System Using Experiments and CFD

Jea-Kyu Park¹, Woo-Duk Kwon², Hyuk-Min Kwon³ and Jeong-Hoon Yang*⁴

¹ Master's Student, Graduate School, Yeungnam University, Korea
² Eco Dream, Korea
³ Doctor's Student, Graduate School, Yeungnam University, Korea
⁴ Professor, School of Architecture, Yeungnam University, Korea

Abstract

This study examines the ventilation function and economic feasibility of a combined air-diffuser ventilation (CAV) system, in which the exhaust air opening and the supply air opening are combined into a single module. A mock-up test and computational fluid dynamics (CFD) were employed to identify the properties of the CAV. The CFD was validated using a benchmark test and then used to investigate the ventilation function of a conventional ventilation system and the CAV using SVE3 and SVE6 for ventilation. The SVE3 and SVE6 results of the CAV were similar to those of the conventional ventilation system. The economic feasibility was assessed based on the duct length and the number of components. In comparison to the conventional ventilation system, the duct length decreased by approximately 68%, the number of dampers decreased by 50%, and the number of diffusers decreased by 40-50% for the CAV.

Keywords: ventilation; CFD; diffuser; indoor air quality

1. Introduction

Apartments in Korea require either natural or mechanical ventilation systems for ventilation exchange of more than 0.5 times per hour¹. In winter, the operation of natural ventilation systems is difficult due to the low outdoor air temperature and the large difference from the indoor temperature. Moreover, energy use is high², and occupants are reluctant to open windows for ventilation. Therefore, mechanical ventilation systems are more effective in that they allow consistent ventilation regardless of the external weather conditions.

For ceiling ventilation systems, the supply and exhaust air openings must be set at an appropriate distance to maintain proper function³⁴. However, longer ducts are needed to maintain this distance. Diffusers in Korean apartments are installed near the walls since lights are installed in the middle of the ceiling. For this reason, there is a high chance of negative drift occurring during ventilation⁵⁶. Therefore, Kwon proposed eliminating the restrictions regarding the diffuser's installation location and using a combined air-diffuser ventilation (CAV) system, which can use a shorter duct⁷⁸. The CAV system combines the supply and exhaust air openings into a single module.

The main purpose of this study is to demonstrate the effectiveness of the CAV. Its economic feasibility was assessed based on its ventilation function and the reduction in the duct length. The focus was on the winter season. CFD was used for the ventilation assessment, and a benchmark test was conducted.

2. Features of the Ventilation System

The concepts of different diffusers for different types of ventilation systems are shown in Fig.1. A conventional ceiling ventilation system is installed at an appropriate distance to prevent any overlap of the diffuser's minimum radius of diffusion. A round diffuser is typically used for the supply and exhaust air openings in Korean apartments. In contrast, the CAV diffuser combines these openings in a single module.

Figs.2. and 3. show examples of the duct design and equipment used for these systems. In the conventional system, the separate supply and exhaust ducts result in an intersection between the ducts, which increase the required ceiling space. In addition, as the room space increases, the length of the duct also increases, and significant static pressure loss occurs. Therefore, the required air ventilation rate cannot be secured for...
each space. However, the combined course of the air supply exhaust ducts in the CAV reduces the length of the duct, the construction time, and costs. Maintenance of the duct is also much easier, and there are fewer restrictions regarding the location.

3. Examination of the Ventilation System

3.1 Experiment Overview

The air velocity and temperature distribution of a conventional system and the CAV were examined experimentally. Two mock-up chambers were installed adjacent to each other in a room (see Fig.4.). The chambers were each 3 m wide, 4 m long, and 2.4 m high, and they were made using an aluminum bar, glass, and Styrofoam insulation to maintain a consistent temperature within them. In addition, all gaps were tightly sealed with non-permeable tape to make the chambers airtight. The experimental conditions were based on the winter season. The floors of the chambers were heated through a heating panel (3.4 m x 1.7 m). The conventional system and the CAV system were installed in Chamber A (right) and Chamber B (left), respectively.

In the conventional system, the supply and exhaust diffusers were installed at a distance of 0.7 m from the walls, while the diffusers of the CAV were installed in the middle of the ceiling. For ventilation of each room, total heat exchangers with the same capacities were installed outside the chambers to supply and exhaust through each diffuser.

Rectangular ducts were used for both systems. The dimensions of the duct cross sections were 110 mm x 55 mm for the conventional system and 160 mm x 40 mm for the CAV. The diffuser was round-shaped in both cases. The diameter was 100φ for the conventional system, while the total diameter was 125φ for the CAV, and the exhaust air opening in the middle was 80φ. The legal basic rate of 0.5 times per hour was secured as the air change rate for each chamber through TAB tests, and the ventilation rate was 15 m³/h.
3.2 Measurement Conditions and Methods
The outdoor air temperature was approximately 
-2.2°C during the experiment. However, the average 
temperature in the room was 9-11°C. The air speed for 
the diffuser and inside the chamber was measured using 
a digital hot-wire anemometer (accuracy of 0.03%, ±0.015 m/s). Fig. 5. shows the measurement points for 
the diffuser’s supply air velocity. In the conventional 
system, A1 and A2 are the measurement points for the 
air speed supplied vertically below the diffuser. A3 is 
the measurement point for when the airflow expands 
indoors due to the guide plate of the diffuser. The CAV 
have a guide plate installed on the exhaust diffuser. B1 
is the point where the air velocity is measured from the 
air that is vertically supplied indoors from the supply 
duct. B2 and B3 are the points where the air speed that 
is horizontally supplied indoors from the guide plate is 
measured.

The air speed was confirmed by measuring for 5 
minutes at each point and calculating the average 
value. The air speed and turbulence intensity were 
measured to determine the boundary conditions of the 
CFD by creating a hole in the supply duct and inserting 
the pitot tube of an anemometer. The turbulence 
intensity was calculated through the equation (1).

\[ T.I = \frac{u'}{U} \]  

(1)

The T.I indicates the turbulence strength (%), \( u' \) 
indicates the standard deviation, and \( U \) indicates 
the average wind speed (m/s).

A T-type thermo-couple was used to measure the 
temperature of the chamber at the points shown in 
Fig. 6. The surface temperatures of the diffusers, walls, 
floor, and ceiling were all measured at 19 points for 
each chamber. The temperatures along the vertical 
direction of the chamber were measured at 5 locations 
that were 0.1, 0.6, 1.1, 1.7, and 2.3 m from the 
ground. These points were also 0.5 m away from a 
corner of the chamber.

A preliminary experiment indicated that the indoor 
temperature reached a steady state at 24-25°C. The 
average value measured for 10 minutes in the steady 
state was used as a boundary condition for CFD. In 
addition, the air speed was also measured when the 
indoor temperature was in a steady state to maintain 
consistency in the experiment.

3.3 CFD Analysis Overview
Many researchers use CFD to assess indoor 
ventilation functions. This approach has been verified 
for its physical reproducibility of real phenomena.\(^\text{13-15}\)

The commercial software ANSYS FLUENT 17.0 
was used for the CFD, and the analysis space is 
shown in Fig. 7. The form and component size of the 
analysis space and ventilation system were modeled 
to be identical to the experiment. The meshes for the 
analysis were divided into coarse, medium, and fine 
levels approximately 820 thousand, one million, and 
1.4 million mesh elements, respectively. The mesh 
elements were tetrahedral cells. To maintain the quality 
of the mesh, the skewness value of all models was 
limited to a maximum of 0.84 and an average of 0.23, 
which are standard values recommended by ANSYS.

The turbulence models used were the Standard 
K-epsilon (SKE) model,\(^\text{13}\) which is typically used for 
airflow analysis; RNG K-epsilon (RKE),\(^\text{14}\) which is a 
modified version of the SKE model; and SST K-omega 
(SKO),\(^\text{15}\) which combines the advantages of the 
Wilcox and K-epsilon models. The SIMPLE algorithm, 
second-order upwind scheme, and surface-to-surface 
radiation method were also used for the analysis. The 
wall function used the enhanced wall treatment.

The boundary conditions of the CFD are as follows.
In the conventional system, the supply air speed of 
the
duct was 0.68 m/s, and the turbulence intensity was 1.3%. For the CAV, these values were set as 0.71 m/s and 1%, respectively. The values achieved through the experiment were used for the air supply temperatures of the ducts, which were 14.6°C and 15.1°C for the conventional system and the CAV, respectively. Heat exchange was generated between the CAV's supply and exhaust ducts as a result of their adjacency. For this reason, the measured air supply temperature was slightly higher than that of the conventional system. The exhaust condition of the duct was set as the outflow.

For the wall of the room in the chamber that was exposed to the air, the average ambient temperature was set as 10.7°C, the heat transmission coefficient was 0.6585 W/m²K, and the thickness was 45 mm. The walls in between the chambers and the ceilings were designated with insulation conditions. Only the side of the aluminum bar exposed to the air was set at the resulting measurement value of 18°C. It was difficult to set clear boundary conditions because there were air cavities in the section of each aluminum bar. As shown in Fig.6., the area of the aluminum bars is much smaller than the entire wall area. To simplify the CFD analysis, the temperature measured with a thermocouple was used for the boundary conditions of the aluminum bars. The temperatures of the heating panel were set as 37.7°C and 38.9°C, respectively.

3.4 Experiment and CFD Analysis Results

In the conventional system, point A1 showed a low air velocity of less than 0.06 m/s. The air speed was 0.74 m/s at A2-1, 0.4-0.5 m/s at A2-2 and A2-4, and 0.24 m/s at A2-3. In the experiment, the guide vane that distributes the airflow equally was not installed on the curved part of the duct connected to the diffuser. Therefore, a pressure difference was generated within this part, and there were differences in air speed between different locations of the diffuser's air supply. A maximum of 3 times the air speed of other directions was generated at A2-1, towards which air is drafted in from the duct.

In the CAV, points B1 and B2 had relatively uniform air velocity in all directions of 0.6-0.88 m/s and 0.3-0.5 m/s, respectively. The decrease in air velocity at B3-1 was slight in comparison to that of B2-1. However, the decrease was greatest (57-62%) at B3-2, B3-3, and B3-4 compared to point B2. Similar to the conventional system, the CAV also produced differences in air velocity in different locations of the diffuser's air supply.

In the CFD analysis, the air speed distributions around the diffusers were similar to the experiment values for the conventional system, regardless of the number of mesh elements and turbulence model. However, a significant pressure distribution was generated near the supply air openings in the CAV because of the narrow openings. All turbulence models produced higher air speed than in the experiment. At point B2, SKO resulted in air speeds that were 0.1-0.3 m/s greater than when using other models. However, the number of mesh elements did not have a significant effect on the analysis results.

As mentioned, the air velocity supplied from the diffuser was not more than 1 m/s in each ventilation system. Accordingly, the environment in the chamber was calm except around the diffuser. The experimental results and the CFD analysis results showed similar air velocities 0–0.2 m/s in the center of the chamber. In the CFD simulation, the air velocity in the center of the chamber did not differ between different numbers of mesh elements and turbulence models (figure omitted).

The vertical temperatures of Chamber A with the conventional system were 24–24.2°C (S.D.: about 0.1). The vertical temperature of Chamber B with the CAV system was 24.2–24.7°C (S.D.: about 0.1). The supply air temperature was 14.5°C (S.D.: 0.17) in Chamber A.
and 15.1°C (S.D.: 0.08) in Chamber B. The exhaust air temperature was 23.6°C (S.D.: 0.1) in Chamber A and 22.8°C (S.D.: 1.06) in Chamber B.

Fig.8. shows the results from the analysis using the experiment's vertical temperature distribution and the fine mesh in CFD. In both the conventional system and the CAV, the CFD results became closer to the experiment results as the number of mesh elements increased. When SKE and RKE were used, the CFD temperature was approximately 0.6-0.9°C higher than in the experiment. In contrast, SKO produced a temperature that was approximately 0.2°C higher, which was the most similar to the experiment results compared to the other models. SKE and RKE showed differences of 0.1-1.0°C in the wall temperatures compared to other models, while SKO showed a difference of about 0.4°C.

The exhaust air temperature of the conventional system was 24.9°C according to SKE, 24.8°C according to RKE, and 24.4°C according to SKO. The exhaust air temperature of the CAV system was 24.0°C according to SKE, 23.8°C according to RKE, and 23.2°C according to SKO. Thus, SKO showed the closest results to the experimental values. SKE is favorable for fully turbulent flow\cite{16}. However, it is not appropriate in the chamber except around the diffuser because the environment there is calm. In addition, it is highly important to analyze the circumference of the wall because the supply air from the diffuser spreads along the wall into the chamber. Thus, RKE with low-Reynolds-number effects\cite{19} was applied to the viscosity, and SKO was applied to the circumference of the wall and the other parts of the chamber\cite{16}. The results of this setup were close to the experimental results. In particular, the SKO results most closely matched the experimental values in the vertical temperature distribution and the exhaust air temperature.

![Fig.9. Conceptual Diagram of Analysis Cases; (a) Case 1, (b) Case 2, (c) Case 3, (d) Case 4, (e) Case 5]

4. Ventilation Function and Economic Feasibility

4.1 CFD Outline

The size of the duct in Section 3 was different for each ventilation system. The air supply temperature of the CAV was also slightly higher than that of the conventional system, since the heat exchange was generated due to the CAV's supply and because the exhaust ducts were adjacent to each other. Heat exchange through the aluminum bar could not be completely blocked since the two chambers were also adjacent to each other. The investigations must be carried out using the same boundary conditions to make an exact comparison between the conventional system and the CAV regarding the ventilation function.

In this analysis space for CFD, the aluminum bar was eliminated as shown in Fig.7. The actual size of the spaces and the composition of the ventilation system are identical to the previous model. In addition, the same boundary conditions were set using the experiment mass flow and temperature of the air supplied to each ventilation system and the temperature of the heating panel of the conventional system. Styrofoam insulation was used for the walls. SKO was used as the turbulence model for the CFD, and the mesh elements were tetrahedral cells. After eliminating the aluminum bar, the number of mesh elements was reduced by approximately 900 thousand.

The CFD analysis cases are shown in Fig.9. Case 1 involves the conventional system. In Cases 2-5, the location, supply, and exhaust directions of the CAV's diffuser were applied differently in each case. In Cases 2 and 3, the diffuser was installed in the middle of the ceiling, while the installation had involved the distancing of the conventional system and the location of Fig.2. in Cases 4 and 5. SVE3 (equation 1) and SVE6 (equation 2) proposed by Kato et al.\cite{16\textsuperscript{17}} were used for the performance assessment of the ventilation system. SVE3 indicates the age of air, and SVE6 indicates the residual lifetime of air.

\[
SVE3(X) = \frac{C'x(X)}{C_S} \quad (2) \quad SVE6(X) = \frac{C''x(X)}{C_o} \quad (3)
\]

SVE3 and SVE6 were analyzed using the passive scalar method after emitting the contaminant \( q \) into the analysis space at a constant rate of 1 kg/s. The scalar fluxes for the walls were set to zero. In this case, the normal airflow was used for SVE3 and the reverse airflow was used for SVE6. The contaminant concentrations at point X in the analysis space are Cx(X) and C''x(X) (kg/s). Cs and Co are contaminant concentrations under the perfect mixing conditions, which can be calculated as the reciprocal of the ventilation rate. Thus, the analysis results of SVE3 and SVE6 can be expressed using the number of air changes per hour (ACH). In SVE3 and SVE6, 1 means 1 ACH, and lower values indicate higher ventilation efficiency.
4.2 CFD Analysis Results

Table 1. shows the indoor air velocity distribution according to CFD and the analysis results of SVE3 and SVE6. In the conventional system (Case 1), low-temperature air was supplied through the supply air opening at approximately 45°C. This was the same as the experiment results. The supplied air is heated by the heating panel, which forms high turbulence along the thermal plume and is released through the exhaust air opening. A congestion zone with low air speed was generated at the center of the room.

In Case 2, the CAV was installed in the middle of the ceiling. The exhaust occurred at the center, and the supply occurred at the side for the ventilation of the CAV. The low-temperature air supplied through the diffuser spread horizontally along the ceiling and then descended down the wall to the floor. The air supplied to the floor was heated by the heating panel and released through the exhaust air opening at the center of the CAV. As a result, turbulence was formed on both sides of the room and generated a congestion zone.

In contrast to Case 2, the supply occurred at the center of the CAV and the exhaust occurred at the side in Case 3. The supplied low-temperature air descended to the floor and destroyed the thermal plume of the heating panel. In Cases 4 and 5, the CAV was installed after applying the distancing of the conventional system. The supply and exhaust directions for the CAV were identical to those of Cases 2 and 3, respectively. In Case 4, low-temperature air was supplied at the

| Case 1 | SVE3 | SVE6 |
|--------|------|------|
|        |      |      |

| Case 2 | SVE3 | SVE6 |
|--------|------|------|
|        |      |      |

| Case 3 | SVE3 | SVE6 |
|--------|------|------|
|        |      |      |

| Case 4 | SVE3 | SVE6 |
|--------|------|------|
|        |      |      |

| Case 5 | SVE3 | SVE6 |
|--------|------|------|
|        |      |      |

Scale

Table 1. Wind Velocity Distribution and Air Age Analysis Results
side of the CAV and formed a strong descending airflow along the wall. In Case 5, low-temperature air was supplied at the center of the CAV, after which a strong, descending airflow formed along the wall. The descending air was heated by the heating panel, rose to the ceiling along the opposite wall, and was released through the exhaust air opening at the side of the CAV. High turbulence formed in the middle of the room during this process.

The air temperature of each chamber was influenced by the airflow and the heating panel (figure omitted). The low-temperature air supplied from the supply air opening was heated by the heating panel, which resulted in a temperature of approximately 24°C in the chamber. The air temperatures in the chamber were similar between all cases.

SVE3 and SVE6 depend on the airflow. The spread of the fresh air supplied from the supply air opening depends on the supply airflow, and low values of SVE3 occurred around the opening. In addition, the air discharged from the exhaust air opening was sucked in according to the airflow. Thus, low SVE6 values occurred around the opening. The SVE3 or SVE6 values did not differ much between each case.

The average values of the occupied zone was calculated for SVE3 and SVE6 for each case in Table 1. and are shown in Fig.10. The chamber's occupied zone was 1.8 m above the floor and 0.3 m from the walls. SVE3 and SVE6 did not differ much between Case 1 (conventional system) and Cases 2-4 (CAV). SVE3 and SVE6 showed similar values in all cases of approximately 0.9 and 1.1, respectively. In other words, the ventilation efficiency was almost identical between both systems. Thus, to prove the validity of the CAV system, it is important to evaluate its economic feasibility with respect to the installation expenses.

4.3 Examination of Economic Feasibility According to the Duct's Length and Used Components

The quantity of ducts and the numbers of dampers and diffusers used in both ventilation systems were calculated based on the 84-m² apartment shown in Fig.2. As shown in Fig.3., the conventional system requires two separate ducts for the supply air and exhaust air. However, the CAV system can be built by partitioning one duct into two portions for the supply air and exhaust air. As a result, the duct length required by the CAV system was 11.8 m, a decrease of approximately 68% compared to the conventional system. Considering the diameter of the ducts, the actual production costs were reduced by approximately 56% with the CAV in comparison to the conventional system.

The damper distributes the air volume. The conventional system uses a T pipe at the junction of the main pipe, and a branch pipe and a splitter damper are used to regulate the air volume and distribute to each room. A volume damper is installed at the bottom of the duct (where the diffuser is located) to regulate the indoor ventilation rate. In contrast, the CAV has a Y pipe at the junction to distribute the airflow in a relatively flexible manner. As the length of the pipes decreases, a volume damper may be installed at only the junction to secure the required air ventilation rate, and the number of installations may also be reduced. Accordingly, the CAV requires less than 50% of the total number of dampers and cost compared to the conventional system.

The conventional system required a total of 12 supply and exhaust diffusers. However, the CAV requires 6-7 diffusers, which is a reduction of 40-50%. The construction cost is also reduced when the number of installations decreases. Thus, the CAV has much higher economic feasibility than the conventional system.

Fig.10. Average SVE3 & SVE6 of Occupied Zone

5. Conclusion

This study assessed the ventilation function and economic feasibility of a CAV system. The results are as follows.

1) A mock-up test was conducted to examine the ventilation function of the conventional system and the CAV. The physical reproducibility and validity of the CFD were verified through a benchmark test of the mock-up test and the CFD. The CFD results using the SKO model and a fine mesh were closest to the experiment values.

2) CFD analysis was conducted using the same boundary conditions to make an exact comparison between the conventional system and the CAV regarding the ventilation function using SVE3 and SVE6 in CFD.

3) SVE3 and SVE6 of the CAV were similar to those of the conventional system. SVE3 was consistent in all cases (approximately 0.9). However, in Case 5 of the CAV, in which the air supply occurs in the middle and the exhaust occurs at the side, there was a slightly low SVE6 in comparison to the other cases, and the exhaust function improved.

4) The duct length was decreased by approximately 68%, the number of dampers was decreased by 50%,
and the number of diffusers was decreased by 40-50% in the CAV compared to the conventional system.

5) The CAV demonstrated similar ventilation functions to those of the conventional system but had better economic feasibility due to the reduced duct length and numbers of components used.

Acknowledgements
This research was supported by the Small and Medium Business Administration of Korea (S2257897).

References
1) Regulation for Facility of the building (Revision 2015). Ministry of Land, Infrastructure and Transport. (In Korea)
2) Energy consumption in Housing and commercial sectors. (2011-2016) Statistics Korea. http://kostat.go.kr
3) Choi, J.M. and Cho, S.W. (2009) A Study on the Indoor Airflow Pattern by Changing the Location of Mechanical Terminal Unit. Korean Journal Of Air-Conditioning and Refrigeration Engineering, 21(3), pp.193-200. (In Korea)
4) Park, J.C. (2006) A Study on the Improvement of Ventilation Effectiveness in High-Rise Apartment Buildings. Journal of the Korean Solar Energy Society, 26(2), pp.67-94. (In Korea)
5) You, B.H., Tanaka, T. and Yoon, C.S. (2002) An Analysis of the Ventilation Efficiency and Airflow Distribution with the Different Types of Furniture Arrangement. Journal of the Architectural Institute of Korea Planning & Design, 18(12), pp.191-198. (In Korea)
6) Moon, H.J., Sohn, J.R. and Hwang, T. (2016) Indoor Air Quality Performance of Ventilation Systems in Classroom. Journal of Asian Architecture and Building Engineering, 15(2), pp.343-348.
7) Kwon, W.D. (2014) Multi function ventilator. Registration No. 20-0474888. Korea Intellectual Property Rights Information Service. http://eng.kipris.or.kr
8) Kwon, W.D. (2016) Ventilation Duct. Registration No. 30-084036. Korea Intellectual Property Rights Information Service. http://eng.kipris.or.kr
9) ISO7726: 1998-Instrument for measuring physical quantities
10) Walker, C., Tan, G. and Glicksman, L. (2011) Reduced-scale building model and numerical investigations to buoyance-driven natural ventilation. Energy and Buildings, 43 (9), pp.2404-2413.
11) Aziz, M.A., Gad, I.A.M., Mohammed, E.S.F.A. and Mohammed, H.M. (2012) Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics. Energy and Buildings, 55, pp.738-746.
12) Seok, H.T., Choi, D.H., Kim, H.J and Yang, J.H. (2009) A Study on the Modeling and Analysis Method according to the Diffuser Shape. Journal of the Architectural Institute of Korea Planning & Design, 26(1), pp.341-348. (In Korea)
13) Lauder, B.E. and Spalding, D.B. (1972) Lectures in Mathematical Models of Turbulence. Academic Press, London, England.
14) Orszag, S.A., Yakhot, V., Flannery, W.S., Boysan, F., Choudhury, D., Maruzewski, J. and Patel, B. (1993) Renormalization Group Modeling and Turbulence Simulations. In International Conference on Near-Wall Turbulence Flows, Tempe, Arizona.
15) Menter, F.R. (1994) Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal 32(8), pp.1598-1605.
16) Kato, S. and Murakami, S. (1998) New ventilation efficiency scales based on spatial distribution of contaminant concentration aided by numerical simulation. ASHRAE Transaction, 94(2): 309-330.
17) Kato, S., Murakami, S. and Kobayashi, H. (1992) New scales for evaluating ventilation efficiency as affected by supply and exhaust opening based on spatial distribution of contaminant. ASHRAE Proceeding of the ISRAVE’92, pp.177-186.
18) Stamou, A. and Katsiris, I. (2006) Verification of CFD model for indoor airflow and heat transfer. Building and Environment, 41(9), pp.1171-1181.
19) Chen, H.J., Moshfegh, B. and Cehlin, M. (2012) Numerical investigation of the flow behavior of an isothermal impinging jet in a room. Building and Environment, 49, pp.154-166.
20) ASHRAE (2009) Handbook-Fundamental, Chapter 20 Space Air Diffusion.