Effect of Design Parameters on Indoor Temperature Distribution in Impinging Jet Ventilated Room

Haruna Yamasawa **Tomohiro Kobayashi** **Toshio Yamanaka**
Member ASHRAE

Narae Choi **Mako Matsuzaki**

ABSTRACT

In many countries, the mixing ventilation system (hereinafter referred to as MV) is the most common and widely used system in office buildings for decades, even after the development of the displacement ventilation system (hereinafter referred to as DV). It is believed that DV has better ventilation efficiency compared to MV. However, DV could have problems such as vertical temperature difference in the occupied zone, horizontal temperature distribution, and performance in the heating mode. To overcome these disadvantages, the impinging jet ventilation system (hereinafter referred to as IJV) has been proposed as a new air distributing system. This work aims to propose a simplified calculation model for IJV. To do this, a parametric study is required, and CFD prediction was chosen as the method to facilitate changing parameters. In the previous study, the authors validated CFD analysis method by conducting both CFD and full-scale experiment, and this paper presents a parametric study. The shape, position, and the number of IJV terminals were changed as parameters. For understanding the effect of the shape of IJV terminals, round and square supply ducts were applied in a room with two terminals. As for the number of terminals, four cases were studied, i.e., one, two, four and six. The position of IJV terminals was also changed in the case with four IJV terminals, and terminals were placed at each corner of the room or centre of each wall. The result shows that the shape and location of IJV terminals does not have a significant influence on mean vertical temperature profile, while the number of IJV terminals strongly affects the velocity and temperature profile. In addition, DR was significantly large when there is only one supply terminal.

INTRODUCTION

The mixing ventilation (MV) system has been the most popular and widely used air conditioning system for decades. However, Boyle (1899) pointed out that in MV, the entire air in the room is mixed and diluted with supplied air to maintain the air quality; therefore, the contaminated air spreads through the room, even to areas that were not originally contaminated. Accordingly, the displacement ventilation (DV) system was introduced to improve the ventilation effectiveness of MV. For example, see Sandberg (1989), Mundt (1995) and Yuan (1998). However, DV also has shortcomings. Since air is supplied with low velocity in this system, there is a possibility that the supplied air rises upwards before it reaches the inner area of the room (see Svensson (1989)). For the same reason, DV cannot be applied for space heating. Additionally, since DV supplies the cooled air directly to the occupied zone, there is an overcooling risk around the ankle level. To overcome the disadvantages of MV and DV, the impinging jet technique (see Rajaratnam (1976)) was applied to the ventilation system as a new air distribution strategy. The impinging jet ventilation (IJV) system is a type of stratification ventilation (see Karimipanah and Awbi (2002), Chen et al. (2012) and Ameen et al. (2019)). The cooled air is supplied downward via a jet to a room, and once it strikes the floor, the supplied air spreads through the room horizontally in a thin layer. If the air velocity is small enough when it reaches the heating sources, the air rises

Haruna Yamasawa is a doctoral course student in the Department of Architectural Engineering, Osaka University, Osaka, Japan. **Tomohiro Kobayashi** is an associated professoet in the Department of Architectural Engineering, Osaka University, Osaka, Japan. **Toshio Yamanaka** is a professor in the Department of Architectural Engineering, Osaka University, Osaka, Japan. **Narae Choi** is an assistant professor in the Department of Architectural Engineering, Osaka University, Osaka, Japan. **Mako Matsuzaki** is a graduate student of the Department of Architectural Engineering, Osaka University, Osaka, Japan.

upward like DV. It is expected that while using IJV, one can achieve a temperature and contaminant concentration similar to that of DV along with a higher momentum. However, it is also expected that the stratification cannot be achieved when the supply momentum is high. The supply momentum varies depending on supply terminal condition even when the total airflow rate is the same. In order to understand the effect of the geometric conditions of supply terminals on indoor environment, CFD analysis was conducted. The shape, positions and the number of terminals were varied as parameter, and the results of analysis are presented in this paper. In addition, since the air supplied to the occupied zone directly with relatively high momentum in IJV, draught risk is also discussed in this paper.

METHODOLOGY

omputational set-ups

The target room studied in this paper is shown in Figure 1. The inner size of the room was 9,000 mm (width) × 5,000 mm (depth) × 2,700 mm (height). For simulating the heat generation from seated human, twenty-four heating elements, whose size was 400 mm (width) × 200 mm (depth) × 1,100 mm (height) with the heat generation of 60 W for each, were distributed inside the room, which equals to 1,440 W as whole room and 32 W/m^2 when averaged through room. The total supply and exhaust flow rate were both 720 m^3/h, and the supply temperature was set to be 20 ºC, so the exhaust temperature is expected to be 26 ºC if the walls were all adiabatic. However, the boundary condition of the walls was not adiabatic. The boundary conditons of walls, floor and ceiling were given as follows: the convective heat transfer coefficient of the external surface was 4.5 [W/m^{2}K], the external temperature was 30 ºC, external and internal emissibity was 0.85. Matsuzaki et al. (2019) validated the method of CFD analysis by comparing the CFD results with that of experiment. The same method of CFD analysis was applied in this study. The analysis was conducted by Ansys Fluent 19.2. The steady state calculation was performed with SIMPLE algorithm under the isothermal condition. SST k-ω model was used as turbulence model, and QUICK scheme was adopted for convective term differencing. Each of the four exhaust openings was located at the centre of the quatre of the ceiling. On the other hand, the positions and number of the supply terminals were varied as the parameter. However, at all cases, the height of the bottom end of the terminals, which is the inlet boundary, were set at FL+600 mm.

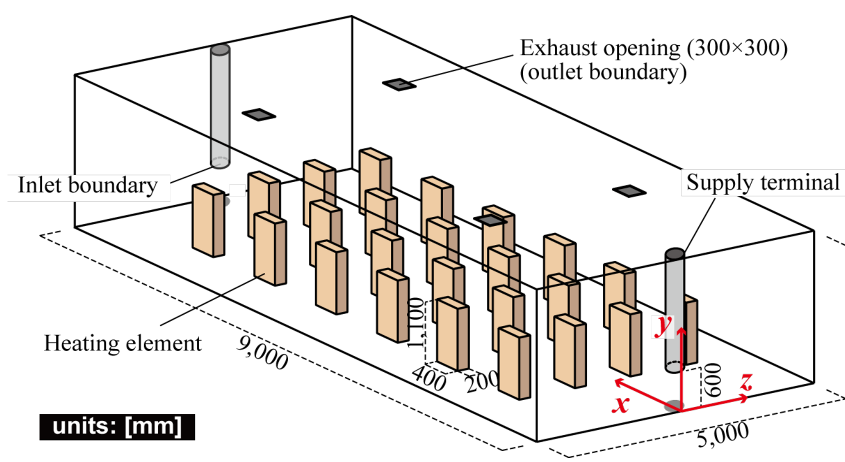

Figure 1 Overview of analytical room (two round ducts).

To understand the effert of the inlet conditions, the shape (round vs square), the number (one, two, four and six) and the positions (corner vs middle of walls) of supply terminals were varied as parameters. The studied cases are summarized in Figure 2. The supply area of the round duct was 225 mmΦ and the square duct was 200 mm × 200 mm. It has to be noted that to decrease computational load, considering the analytical room was symmetric, the analytical domain was the half of the room in the case with one supply terminal, and was the quarter in other cases.

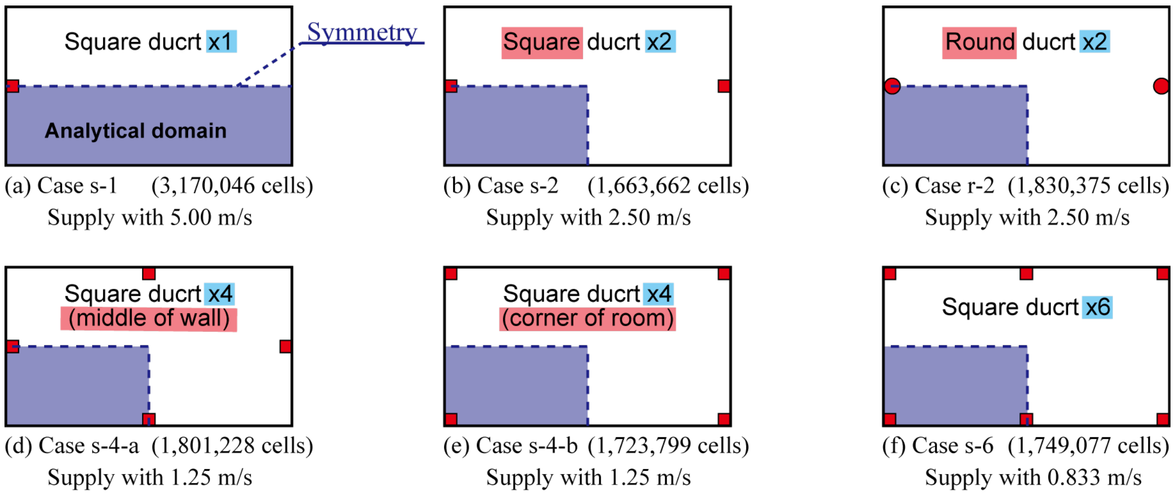

Figure 2 Summary of terminals status and analytical domain.

Definition of index

Since the cooled air is supplied to the lower part of room directly, it might cause the unpleasant cold draught to the occupant. Thus, the evaluation of draught risk is important for IJV. Fanger et al. (1988) proposed an equasion for predicting percentage of dissatisfied people caused by draught risk as follows:

$$DR = \left(T_{skin} - T_p\right)\left(U_p - 0.05\right)^{0.6223}\left(0.3696 U_p I_p + 3.143\right) \tag{1}$$

where, T_{skin} is the averaged temperature of skin (34 °C), T_p is the rremperature at each point, U_p is the flow velocity at each point and I_p is the turbulent intensity at each point. The applicable range of the equasion is: $0 < U_p < 0.4$ [m/s], $20 < T_p < 26$ [ºC] and $0 < I_p < 70$ [%]. It has to be noted that even if U_p was smaller than 0.05 m/s, the the equasion should be caluculated with $U_p = 0.05$ m/s, and even if DR exceeded 100%, the results shoud be considered as 100%. DR within occupied zone is recommended to be not exceed 15%. Since U_p is assumed to be time averaged velocity magnitude, which is obtained by omnidirectional probe, Reynolds-averaged velocity $\overline{V}$ [m/s] of CFD results is converted based on Popiolek's equation (Popilek and Melikov (2008)) shown below:

$$U_p = \begin{cases} \overline{V}(1 - 0.044\, I + 1.195\, I^2 - 0.329\, I^3) & if \quad I \leq 130\% \\ \overline{V}(0.287 + 1.502\, I) & if \quad I > 130\% \end{cases} \tag{2}$$

RESULTS AND DISCUSSIONS

Temperature and flow velocity distribution

The analytical results of the cases with two terminals with different inlet shapes are compared in Figure 3 (a). It was shown that even though the shapes of duct were different, the distributions of both temperature and flow velocity were almost the same. The results of four terminals with different positions are compared in Figure 3 (b). It seems that the temperature distribution of the case with ducts located at the middle of walls was more horizontally uniform than that of the case with ducts located at the corner, however, the difference was negligible.

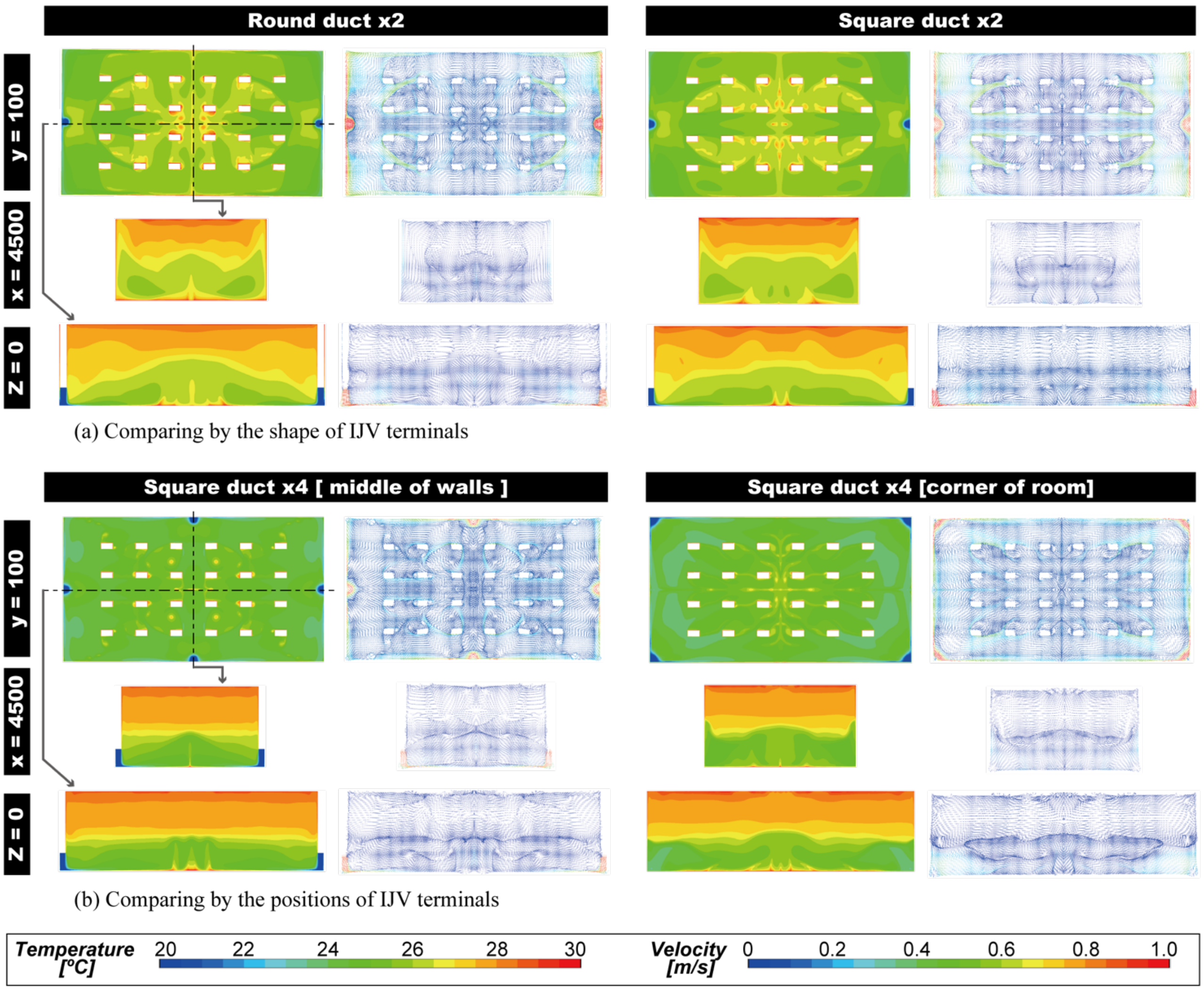

Figure 3 Comparison of temperature and velocity distributions by the shape and positions of supply terminals.

The analytical results of the cases with one terminal and six terminals are compared in Figure 4. It was shown that the temperature distribution was not stratified vertically in the case with one supply terminal, while there was temperature stratification in other cases. By comparing all of the cases, it was shown that by increasing the number of teminals, the temperature tends to stratify clearly. The results of four terminals and six terminals were almost the same, thus, it was found that there is sufficient number of terminals for achieving uniform horizontal temperature distribution. Additionally, in the cases with small number of supply terminals, the velocity was relatively large, which might cause the cold draught to the occupants.

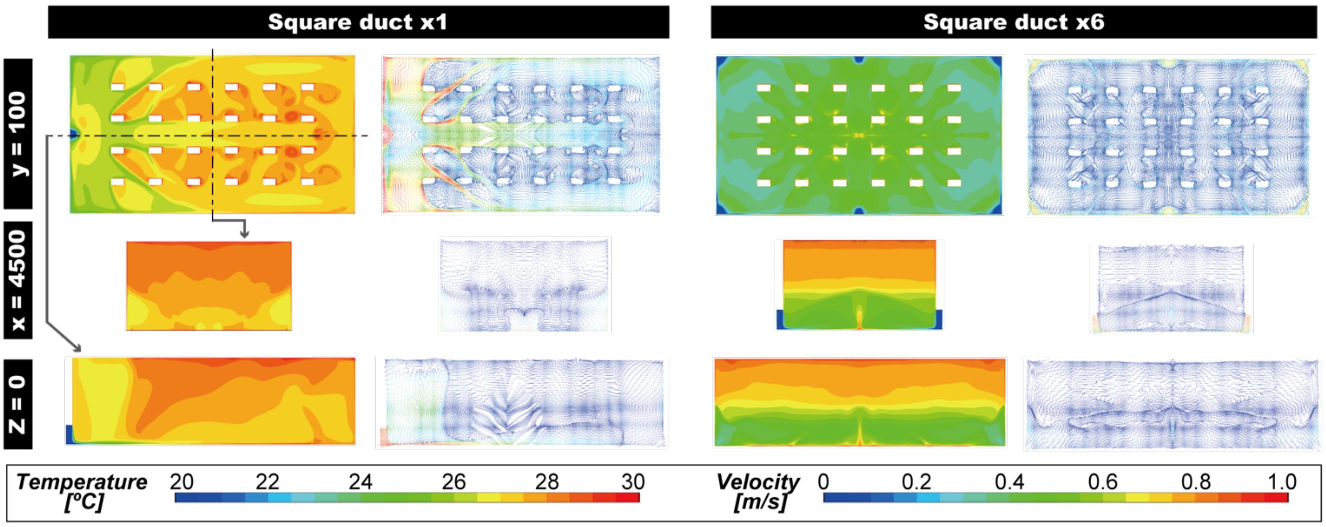

Figure 4 Comparison emperature and velocity distributions by the number of supply terminals (one vs six).

Horizontal temperature distribution

The temperature at the evalution points shown in Figure 5 (Left) was averaged at each height and are shown as the vertical temperature distribution in Figure 5 (Right). As it was shown in previous section, the horizontal distributions of temperature were almost the same even the shape of the terminals was different, and there was slight difference when the positions of terminals were different. However, when the results were horizontally averaged, the vertical distributions were almost the same in both cases. In addition, the vertical temperature distiribution of the case with one terminal was relatively uniform, and that of the case with two terminals was relatively uniform at the lower level of the room. However, by increasing the number of terminals, it was found that a clear vertical temperature stratification is formed. Therefore, shape and position of IJV terminal doess not have large impact on temperature distribution, while the number of terminals, i.e., supply velocoity, significantly affects indoor temperature distribution.

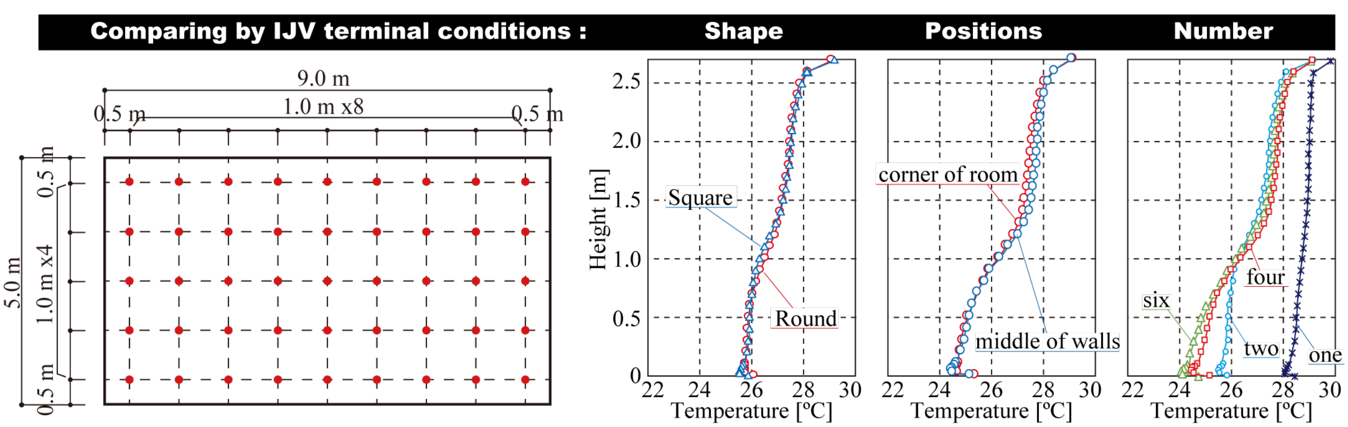

Figure 5 Comparison of vertical distributions of horizontal-averaged room temperature (Left: Evaluatoin points of temperature (Plan), Right: Numerical results).

Evaluation of draught

The DR distributions of (a) the cases with two terminals with different inlet shapes and (b) the cases with four terminals with different positions of reminals are compared in Figure 7. It was found that the draught risk is high around the supply terminals and within the thin layer of supply jet close to the floor. As it shown on the cross section at z = 0 mm, DR was 25 to 30 % within the developing region of the wall jet along the floor, however, at the height of 100 mm, DR was smaller than 15 % at most of the area, which meets the recommended value. Moreover, DR was smaller than 10 % at most of the area at the height of 600 mm. In addition, by changing the shape or position of supply teminals, the distribution of high DR area differed, especially at lower level.

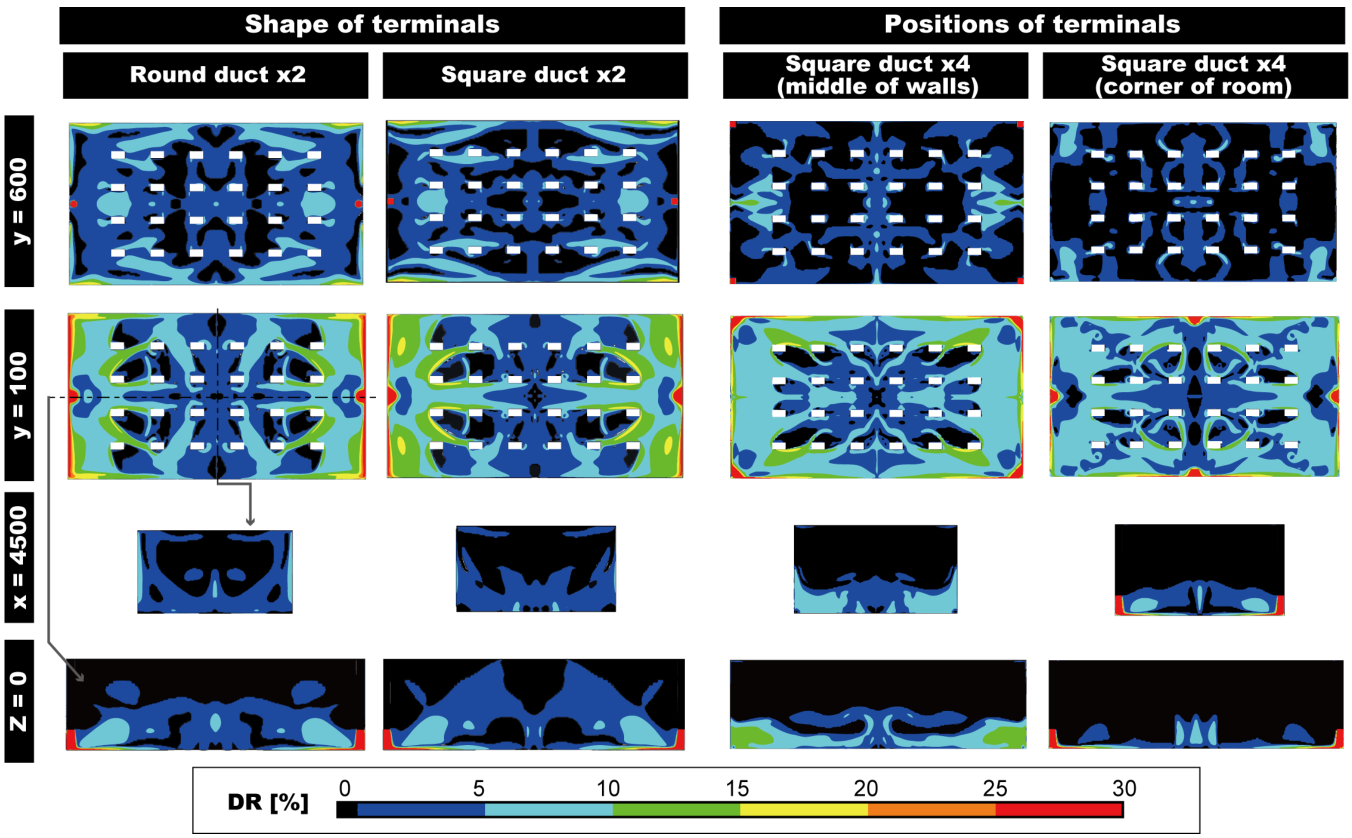

Figure 7 Comparison of DR by shape and positions of supply terminals. (the black area indicates Up<0.05 m/s, i.e., DR equals to 0)

The DR distributions of the cases with different number of terminals are compared in Figure 8. When there was only one supply terminal, DR was more than 15 % around the supply terminal, even at the height of 100 mm. In the previous section, the vertical temperature distributions were almost the same at the cases with four terminals and six terminals, but the results of DR were better at the cases with six terminals, especially at the lower level of the room. However, DR was smaller than 15 % at most of the area inside the room in the case with four terminals. It was found that the number of terminals, i.e., the supply velocity has larger influence on DR than shape and position of terminals.

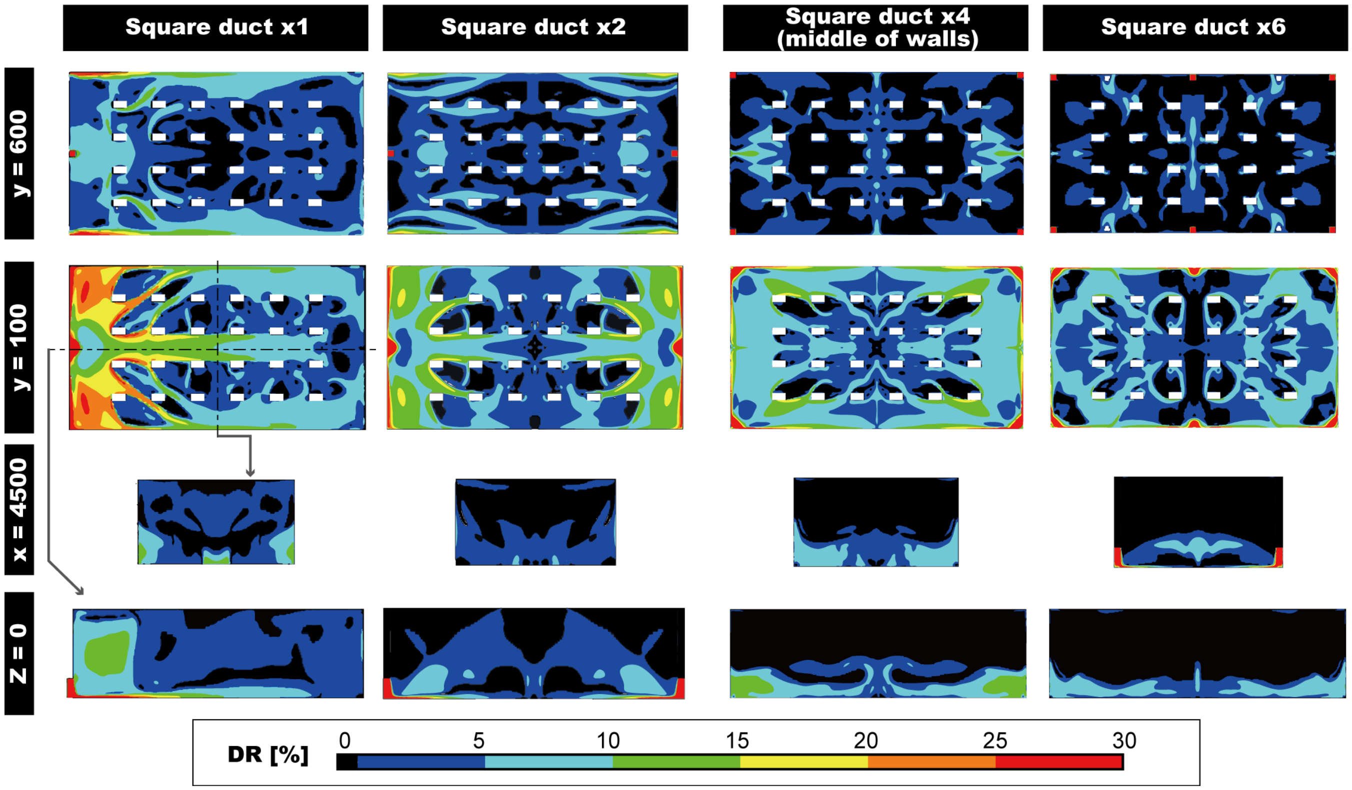

Figure 8 Comparison of DR by number of supply terminals (the black area indicates Up<0.05 m/s, i.e., DR equals to 0).

CONCLUSION

In order to understand the effect of design parameter of supply terminals on indoor environment, CFD analysis was conducted. The shape, the position and the number of terminals were changed as parameter. For the evaluation of draught risk, DR distribution was also evaluated. The findings obtained in this paper are summarized follows:

- By comparing the results of the cases with different shape of terminals, it was shown that the horizontal distributions of temperature and velocity were almost the same, however, DR was slightly different.
- By comparing the results of the cases with different positions of terminals, it was shown that the horizontal distributions of temperature, velocity and DR were slightly different, but the difference was not very large.
- By changing the number of terminals, the results varied significantly. The number of terminals has a large effect on the temperature, velocity and DR distribution, compared to shape and the positions of terminals.
- As the number of terminals increases, the horizontal distribution of termperature and velocity became more uniform, and the termperature stratified more clearly.
- When there was only one supply terminal, the horizontal temperature and velocity distribution were not uniform, and the vertical termperature distribution was not stratified.
- DR was noticeably large in the vicinity of floor. However, because the supply jet spread through the room with thin layer, DR was lower than 15 % at most of the area at FL +100 mm, which is the recommended value, when there were more than two supply terminals.

The temperature and velocity distributions were evaluated in this paper; however, further study has to be done about the contaminant removal. Additionally, the parametric studies about heating also should be done.

ACKNOWLEDGMENTS

This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

NOMENCLATURE

I = Turbulent intensity [-]
DR = percentage of dissatisfied by draught [%]
T = Temperature [°C]
U = Flow velocity [m/s]_

Subscripts

p = at each point
$skin$ = the averaged value of skin

REFERENCES

A. Ameen, M. Cehlin, U. Larsson, T. Karimipanah, Experimental investigation of the ventilation performance of different air distribution systems in an office environment—cooling mode, Energies. 12 (2019) 1354–1368. https://doi.org/10.3390/en12071354.

A.G.L. Svensson, Nordic experience of displacement ventilation systems, ASHRAE Trans. 95 (1989) 1013–1017.

E. Mundt, Displacement Ventilation Systems- Convection Flows and Temperature Gradients, Build. Environ. 30 (1995) 129–133. https://doi.org/10.1016/0360-1323(94)E0002-9.

H.J. Chen, B. Moshfegh, M. Cehlin, Numerical investigation of the flow behavior of an isothermal impinging jet in a room, Build. Environ. 49 (2012) 154–166. https://doi.org/10.1016/j.buildenv.2011.09.027.

M. Matsuzaki, T. Kobayashi, T. Yamanaka, N. Choi, H. Yamasawa, Vertical Distribution of Temperature and Contaminant Concentration in a Room with Impinging Jet Ventilation System AIVC 2020 Proc., 2019

M. Sandberg, C. Blomqvist, Displacement ventilation systems in office rooms, ASHRAE Trans. 95 (1989) 1041–1049.

N. Rajaratnam, Turbulent Jet, Elsevier Science Ltd., Amsterdam, 1976.

P.O. Fanger, A.K. Melikov, H. Hanzawa, J. Ring, Air turbulence and sensation of draught, Energy Build. 12 (1988) 21–39. https://doi.org/10.1016/0378-7788(88)90053-9.

O. Boyle, Natural and Artificial Methods of Ventilation, Nature. 61 (1899). https://doi.org/10.1038/061006a0.

T. Karimipanah, H.B. Awbi, Theoretical and experimental investigation of impinging jet ventilation and comparison with wall displacement ventilation, Build. Environ. 37 (2002) 1329–1342. https://doi.org/10.1016/S0360-1323(01)00117-2.

X. Yuan, Q. Chen, L.R. Glicksman, A critical review of displacement ventilation, ASHRAE Trans. (1998) 78–90.

Z. Popiolek, A.K. Melikov, D. Removal, Improvement of CFD predictions of air speed turbulence intensity and draught discomfort, in: Indoor Environ., 2008: pp. 17–22.