Ventilation and Thermal Performance Examination of Slot Line Diffuser for Perimeter Usage by CFD Simulation

Shaoyu Sheng*[1], Toshio Yamanaka[1], and Tomohiro Kobayashi[1]

1 Osaka University
2-1 Yamadaoka
Suita, Osaka, Japan
**Corresponding author:*
sheng_shaoyu@arch.eng.osaka-u.ac.jp

ABSTRACT

Building ventilation demand increased during the “new normal” following the Covid-19 pandemic. Rather than completely renovating existing HVAC equipment, it is more practical and cost-effective to maximize their existing ventilation performance. This study focuses on slot line diffusers, which are widely used for handling perimeter heat loads. Its long throw distance and wide coverage effectively block cold drafts and heat flow from the exterior window glass. Meanwhile, the mounting position near the glass surface and the high supply velocity can create a ventilation scenario similar to wall confluent jet ventilation (WCJV) or ceiling-supplied displacement ventilation. In order to determine the ventilation and thermal performance of the slot line diffuser, CFD simulations have been conducted using the low-Reynolds number turbulence model with fine meshes, in which a detailed velocity characteristics model derived from a full-scale experiment is used to reproduce the diffuser’s supply airflow. This study priority examined the performance under heating supply conditions. This is because buoyance reduces the throw distance of heating airflow, which obviously influences thermal and ventilation performance compared with cooling conditions. The analysis space is modelled based on a typical open space between two columns of an office building in Japan. Four human simulators are seated around a table in this space’s interior to reproduce a small-scale meeting scenario. Gas is generated by a human simulator to represent pollutants (odor or droplet nuclei) from a speaker. The diffuser’s air supply mode (full-open and half-open) and the mounting location of the exhaust are adjusted as parameters. The CFD simulation results provide a comparison of the ventilation and perimeter heating performance of this type of air terminal by listing 1: the distribution of tracer gas concentration, air temperature, and airflow distribution in the occupancy zone; 2: heating capacity, air distribution performance index (ADPI), draft rate (DR), and ventilation efficiency under different use scenarios. In conclusion, decreasing the slot line diffuser’s supply area obviously improves its heating efficiency by blocking the cold draft from the exterior window. It also enhances ventilation performance by establishing a WCJV. The vertical temperature difference and average normalized concentration in the occupied zone can be kept below 1°C and around 0.7, respectively, for office heating usage in this study.

KEYWORDS

Slot line diffuser, Perimeter thermal environment, Wall confluent ventilation, CFD simulation

1 INTRODUCTION

From the view of maximizing the ventilation and thermal performance of existing HVAC equipment, this study concentrates on the slot line diffuser, an air terminal that is widely used for perimeter heat treatment in Japan. However, studies on the slot line diffuser are mainly concentrated on its interior ventilation performance as used for stratum ventilation [1], protected zone ventilation [2], stratified air conditioning [3], etc. Regarding its perimeter usage, Lorch et al. [4] examined the perimeter air conditioning performance by mock-up tests, and Rousseau [5] tested the Air-Diffusion Performance Index (ADPI) by a full-scale experiment. Except for these two studies conducted in the early 1980s, no further studies have been found using the new measurement or simulation method in recent years, and investigation on its perimeter ventilation performance remains inadequate. Hence, our study will examine both the thermal and ventilation performance of the slot line diffuser mounted in the perimeter area with large glazing areas.

Designed with a slim geometric shape, the slot line diffuser jets air from a wide range and works as an air curtain. At the same time, a wide supply area decreases outlet velocity. This means less momentum, and buoyancy will obviously reduce the throw distance of the heating air supply, reducing heating efficiency during winter. Furthermore, this type of terminal is expected to establish wall confluent jet ventilation (WCJV) [6] or ceiling-supplied displaced ventilation to enhance ventilation efficiency. However, insufficient throw distance may have adverse effects. Therefore, in our research, two deflection panels are installed inside the diffuser's chamber to halve the diffuser's outlet area if needed. Their effect on converging airflow has been proved in our previous study [7], and their impact on the perimeter thermal environment and ventilation performance under the heating supplied condition will be examined by detailed computational fluid dynamics (CFD) simulation in this study.

In order to ensure CFD simulation accuracy, a detailed model for reproducing supply air from this type of terminal will be used. Based on the full-scale experiment in our previous study [7], this model was proposed and validated to have greater precision than the P.V. method or simple condition method. Furthermore, a linear low-Reynolds-number turbulence model [8] [9] with fine meshes will be applied. Compared with the commonly used Standard k-ε (SKE) turbulence model for indoor environment simulation, this turbulence model could predict the attached and separated wall shear flows and have higher accuracy in calculating the convection heat transfer. Tracer gas will be generated from a human simulator's mouth. Ventilation performance will be assessed by examining the distribution of gas concentrations in the analysis space. Furthermore, the distribution of the flow field and the age of air in this space will also be exhibited in graphs. With respect to the perimeter thermal performance, the temperature and airflow distribution in the analysis space, heat transfer rate and temperature on the glass's inner surface, heat removal rate in the perimeter zone, heat transfer/exchange rate between the perimeter and interior, and the air distribution performance index (ADPI) and draft rate (DR) index in the analysis space are enumerated. The geometry of the outlet surface in the longitudinal direction (deflection panels adjustment), and the exhaust's mounting location are adjusted as parameters. By comparing the simulation and calculation results, the thermal performance of the slot line diffuser in the glazing area during the heating period is determined. In addition, the possibility of its use as a ventilation system will be examined.

2 DETAILS OF THE CFD SIMULATION

2.1 Slot line diffuser and the detailed velocity characteristics model

The archetype of the slot line diffuser with adjustable supply area examined in this study shows in ***Fig.1***. As shown in ***Fig.1(a)***, closing these panels converged the airflow, thereby halving the diffuser's supply area into 550 mm longitudinal (hereinafter abbreviated as 1/2 outlet mode), as well as if the panels are opening, the supply area is 1200 mm longitudinal (hereinafter abbreviated as 1/1 outlet mode). Based on the experiment and CFD simulation in the previous study [7], under the 200m^3/h, the standard supply rate of this diffuser, 1/2 outlet mode can throw the supply air into the occupied zone from the 3 meters height ceiling at a speed above 1.0 m/s. And under the same supply rate, outlet airflow of the 1/1 mode has a reversed "V" shape distribution, extending the diffusion range in the longitudinal direction, but airflow velocity in the occupied zone is less than 0.3 m/s.

Fig.2 briefly shows the airflow characteristics model and the detailed boundary division method proposed in our previous study. Each airflow boundary is defined as a "fixed velocity opening", having its unique component velocity profile in *u, v, and w* components calculated from the "airflow characteristics model" based on the full-scale measurement data using the X-type hotwire anemometer. Furthermore, the velocity data have also been used to calculate the turbulence kinetic *k* and dissipation *ε*. Mean values of *k* and *ε* calculated from every measurement point are used to define the turbulence statistics data under each air supply mode, respectively.

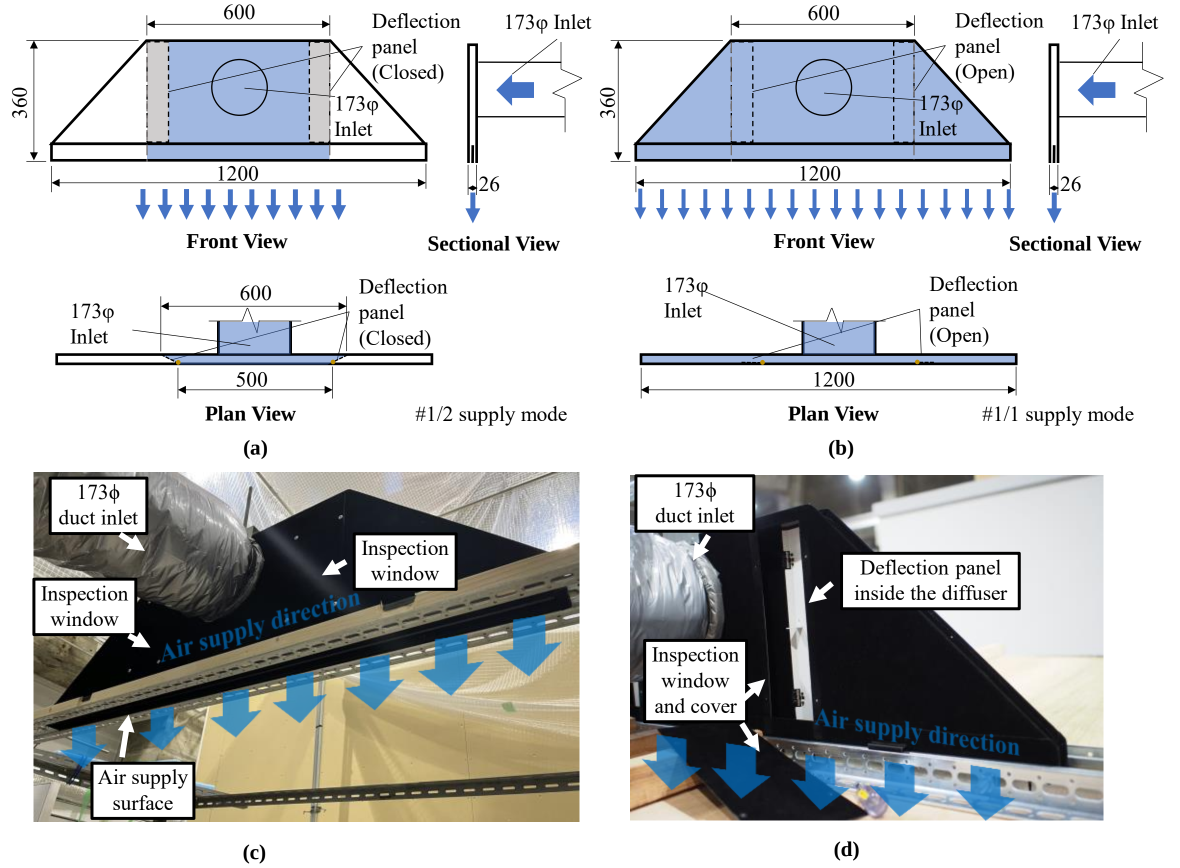


Figure 1 Slot line diffuser with deflection panel for supply area adjustment; (a) image of the 1/2 outlet mode; (b) image of the 1/1 outlet mode; (c) exterior photo of the research subject; (d) detailed view of the panel

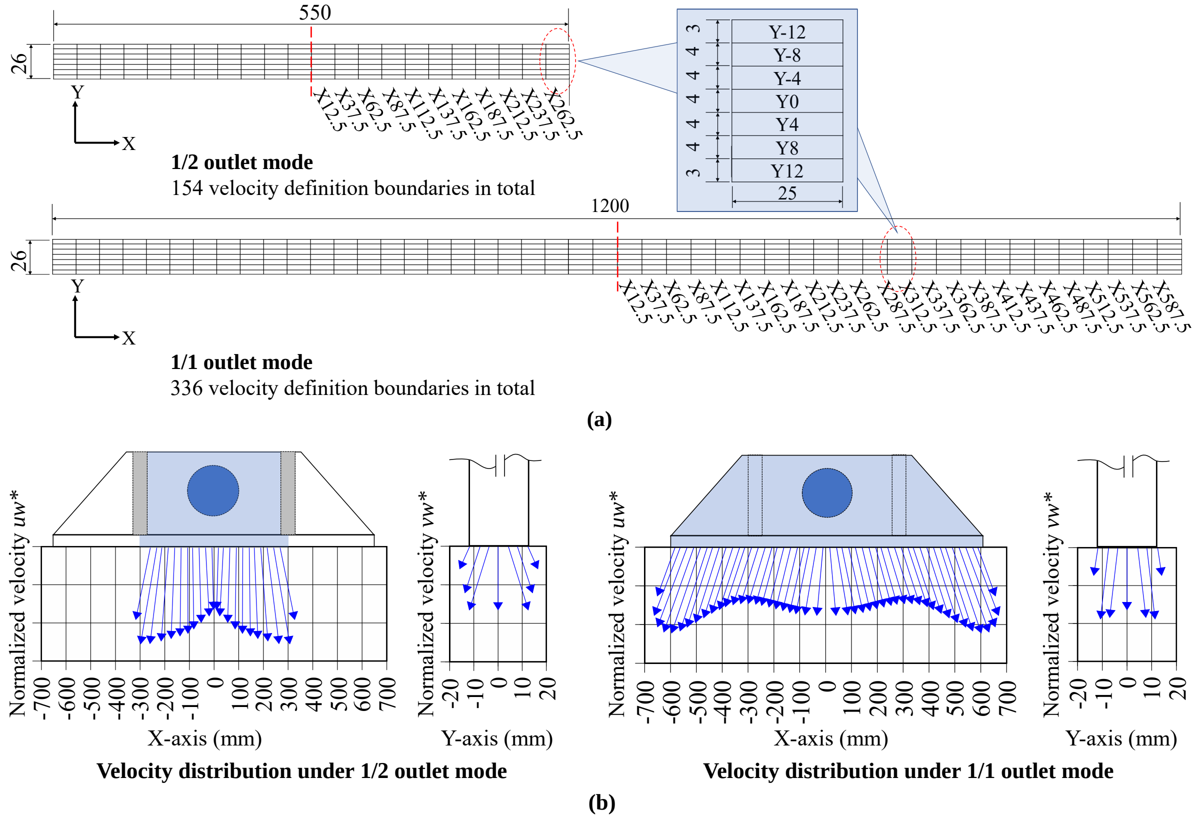


Figure 2 Airflow boundaries and the supply velocity distribution of the slot line diffuser for CFD usage; (a) layout of the velocity and turbulence specify boundaries for two supply modes; (b) normalized velocity distribution of the *uw* and *vw* component for boundaries' velocity calculation

2.2 CFD analysis space and the parameters

Fig.3 shows an overview of the analysis space in this study's CFD simulation. ***Table 1*** shows the CFD's parameter, and Table 1 summarizes the details of the CFD simulation. The dimension of this analysis space is modeled based on a typical open space between two columns of an office building in Japan. This space has 6 mm thick single quartz glass (thermal conductivity K=1W(m·K) on one side, considered a glass facade with poor insulation to examine the diffuser's perimeter performance. And the OA temperature is defined as 2°C, considering Japan's winter climate. The inner surface of the quartz glass is a no-slip wall and conduction heat transfer boundary due to applying the low-Renoleyd number turbulence model. Four human simulators with 75W heat generation sit in the room's interior and surround a table. Tracer gas has the same physical property as air are generate from the mouth (1.1m height from the floor) of one human simulator. Details about the heat and gas generation are shown in ***Table 2***.

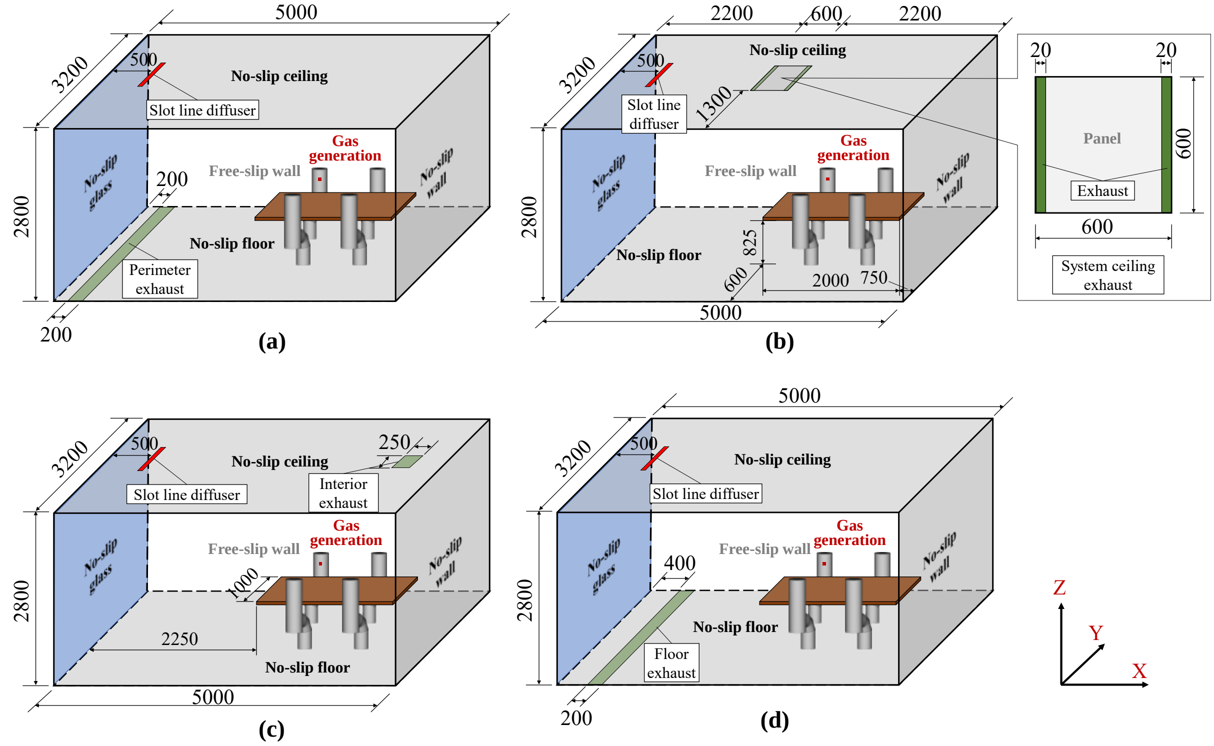


Figure 3 Image of the CFD analysis space with a slot line diffuser in perimeter glazing area; (a) perimeter floor exhaust (0.2m away from the glass); (b) system ceiling exhaust; (c) interior ceiling fan exhaust; (d) floor exhaust 0.4m away from the glass)

Table 2: Case number, abbreviation, and parameters of the CFD simulation

Cases	Parameter abbreviation	Supply mode	Supply air	Exhaust
Case1	05m (1/2) PerE_30	1/2	$200m^3/h$, 30°C	Floor (0.2m from the window)
Case2	05m (1/2) CeilE_30			System ceiling (room's center)
Case3	05m (1/2) IntE_30			Ceiling fan (interior ceiling)
Case4	05m (1/2) FlrE_30			Floor (0.4m from the window)
Case5	05m (1/1) PerE_30	1/1		Floor (0.2m from the window)
Case6	05m (1/1) CeiE_30			System ceiling (room's center)
Case7	05m (1/1) IntE_30			Ceiling fan (interior ceiling)
Case8	05m (1/1) FlrE_30			Floor (0.4m from the window)

The supply boundary of the slot line diffuser is mounted on the ceiling and 0.5m away from the glass's inner surface. In addition, four kinds of exhaust methods are used to examine their influence on perimeter environments. The first one is the 3200 mm (W) and 200 mm (L) perimeter floor exhaust, mounting on the floor 200 mm away from the window glass (***Fig. 3***

(a)). The second is the exhaust in the system ceiling generally used in Japan, and its enlarged view shows in ***Fig. 3 (b)***. The third is the interior exhaust shown in ***Fig. 3 (c)***, which is a 250mm square considered a ceiling exhaust fan. The last is a floor exhaust with the same scale as the perimeter floor exhaust but located 400mm from the window glass ***Fig. 3 (d)***. Furthermore, all these three kinds of exhausts are defined as the natural outflow boundary.

Table 1: *CFD settings and conditions used in this study*

Analysis Method	CFD code		STREAM V2022
	Turbulence model		Linear low-Reynolds-number k-ε model
	Algorithm		SIMPLE
	Radiation analyzed		VF method
	Discretization scheme		QUICK
	Number of calculation cycle		5,000
Meshing	Number of mesh		9,200,000
	Standard mesh size		25 mm
	Standard geometric ratio		1.1x
	First mesh from glass/wall surface	Perimeter glass	0.2 mm thickness
		Wall, ceiling, floor	0.5 mm thickness
	6 mm single quartz glass		Six divisions in the thickness direction
Glass and wall	Perimeter glass	External surface	Specifying heating transfer coefficient, h =23W(m^2·K), OA =2°C
		Material	6 mm single quartz glass; K=1W(m·K)
		Inner surface	No-slip wall boundary, Conduction, Emissivity specify =0.9
	Ceiling, floor, and interior wall		No-slip wall boundary, Conduction, Temperature specify =22°C, Emissivity specify =0.9
	Partition wall		Free-slip wall boundary, Temperature specify =22°C, Emissivity specify=0.9
Airflow boundary	Slot line diffuser	1/1 outlet mode	550 mm (L) × 26 mm (W),154 inlet boundaries; Fixed velocity (around 4m/s); k and ε specify Supply air: 200m^3/h, 30°C
		1/2 outlet mode	1200 mm (L) × 26 mm (W), 336 inlet boundaries; Fixed velocity (around 1.8m/s); k and ε specify Supply air: 200m^3/h, 30°C
	Exhaust	Perimeter exhaust	3200 mm (L) × 200 mm (W), Natural outflow
		Interior exhaust	3200 mm (L) × 200 mm (W), Natural outflow
		Ceiling exhaust	600 mm (L) × 20 mm (W) × 2 slots, Natural outflow
Furniture and occupants	Dimension	Human simulators	300 mm diameter cylinder in sitting position × 4
		Table	2000 mm (L) × 1000 mm (W) × 50 mm (H)
	Heat generation	Human simulator	75W × 4= 300W in total
	Tracer gas generation	Mouth size	14 mm (L) × 13 mm (W)
		Tracer gas	9.95L/min; 34°C; Physical properties same as air

2.3 The Low-Reynolds Number *k*-*ε* Model and the Meshing

The linear low-Reynolds-number model suggested by Abe., et al. [8] [9], was used in this study's turbulence calculation. The linear low-Reynolds number k-ε model was confirmed to be capable of predicting the attached and separated wall shear flows in several studies and was considered suitable for this study's perimeter heat transfer calculation. A detailed mesh division is required to reproduce turbulence statistics near the wall in this model. Therefore, the mesh division of the case with 1/2 supply & interior floor exhaust is shown in ***Fig. 4*** for reference. All the simulation cases used the same mesh division near the glass and wall to avoid the influence of meshing dependency. The dimensionless wall distance y+ from the glazing inner surface is ensured to be less than 0.5.

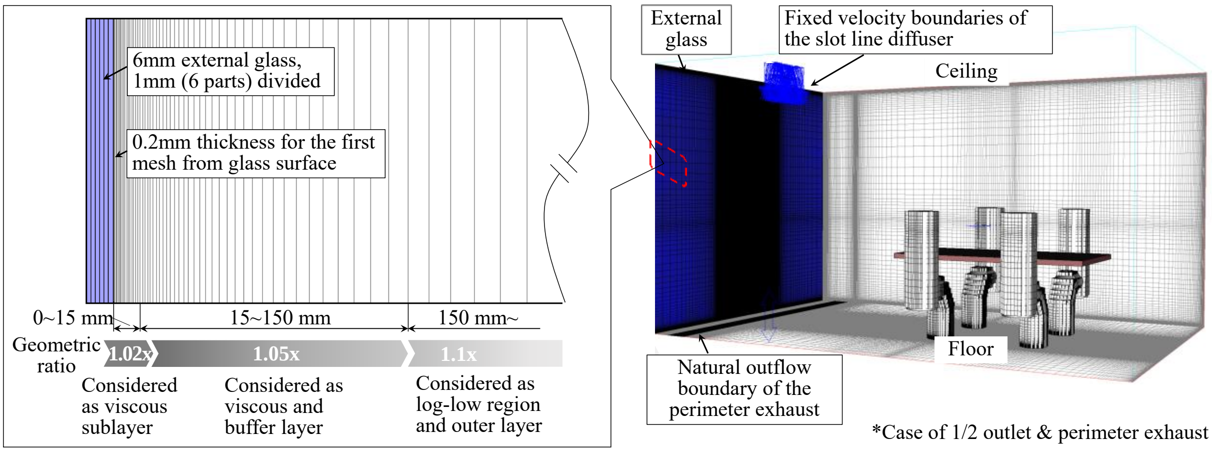


Figure 4 Image of the mesh design in the whole analysis space and the mesh division in the space near the glass surface

3 RESULTS

3.1 Thermal performance

The contour graphs of indoor air temperature and scalar velocity distribution on the vertical section of the room's longitudinal center, and the horizontal section 0.1 m above the floor are shown in ***Fig.7*** and ***Fig.9*** (diffuser's 1/1 and 1/2 supply mode, respectively). The isosurface of velocity = 0.3 m/s, the exhaust temperature, the air conditioning system's heating capacity q_{AC} , the average temperature in the occupied zone, and the glass's inner surface temperature are also summarized in these two graphs for reference.

The mean data of the air temperature, air distribution performance index (ADPI), and draft rate (DR) index shows in ***Fig.5***; the vertical temperature and scalar wind speed distribution in the occupied zone shows in ***Fig.6*** for comparison.

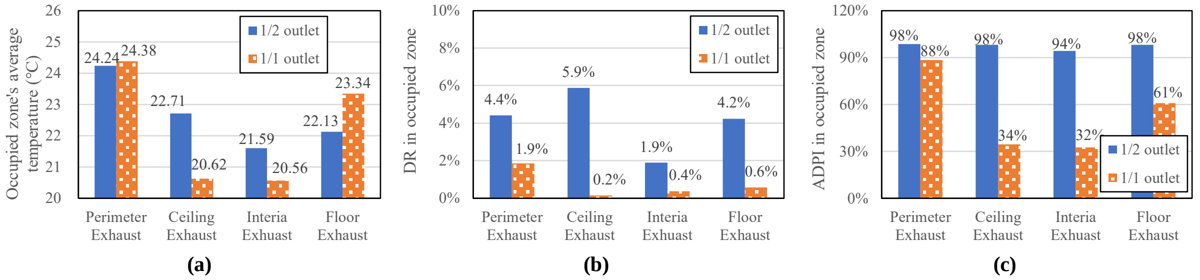


Figure 5 The ADPI, DR, and air temperature's relationship with the supply/exhaust methods (a) occupied zone's ADPI; (b) occupied zone's DR; (c) occupied zone's average temperature

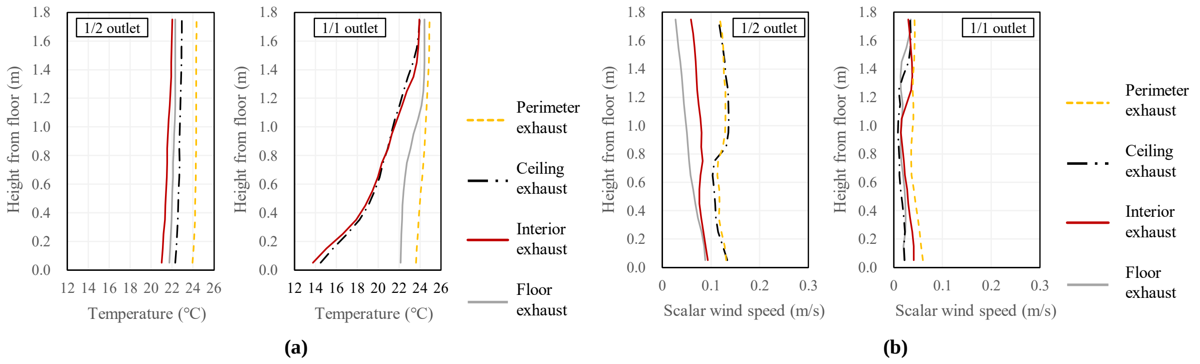


Figure 6 The air temperature and airflow vertical distribution's relationship with the supply/exhaust methods; (a) temperature's vertical distribution in the occupied zone (average of the data with the same height coordinate); (b) scalar win speed's vertical distribution in the occupied zone (average of the data with the same height coordinate)

	05m (1/1) PerE_30	05m (1/1) CeilE_30	05m (1/1) Int_30	05m (1/1) FlrE_30
Exhaust temperature (°C)	22.18	24.05	24.10	22.01
AC's capacity (W)	509.03	387.62	384.39	520.56
Average air temperature in the occupied zone (°C)	24.38	20.62	20.56	23.34
Average temperature of glass inner surface (°C)	8.64	8.62	8.56	8.68

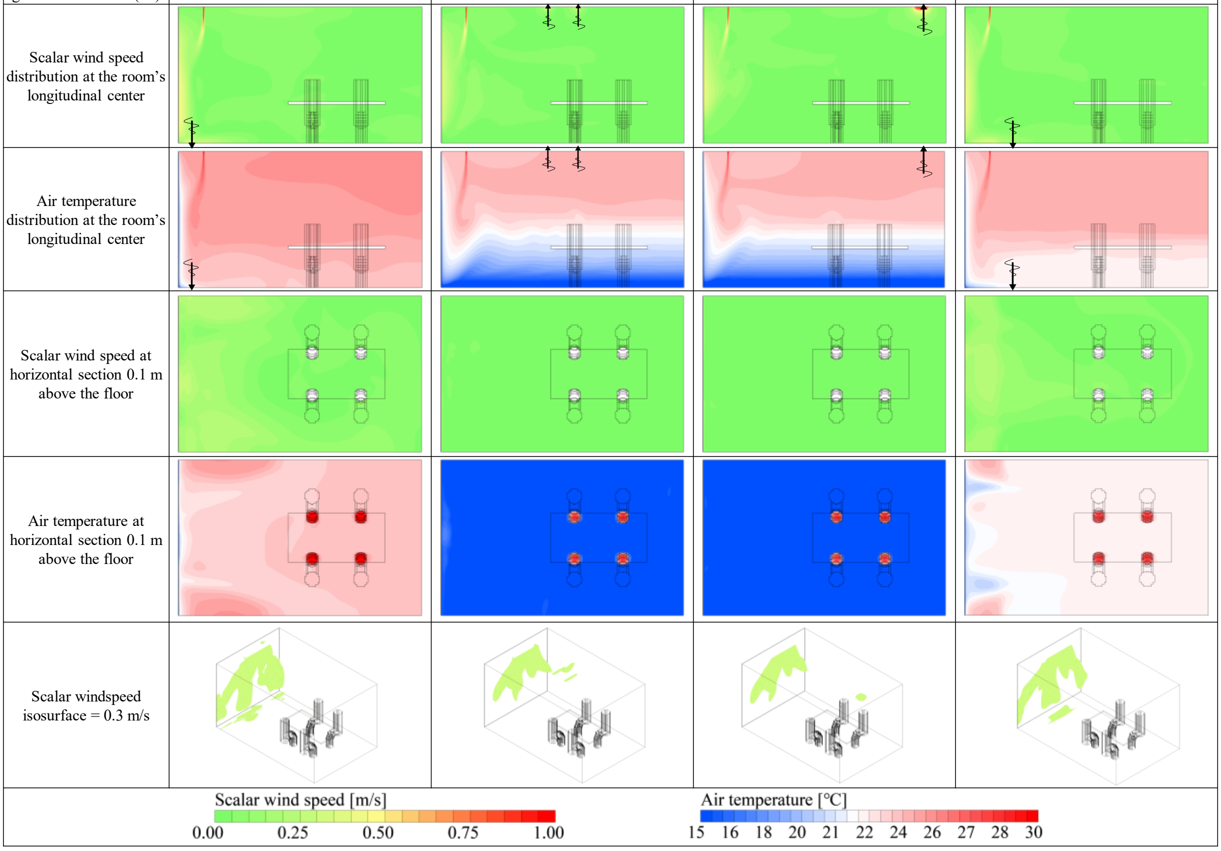


Figure 7 1/1 outlet mode of the slot line diffuser; air temperature and scalar wind speed distribution in the analysis space while using the different exhaust methods (perimeter floor, ceiling's center, interior's ceiling, and the floor exhaust)

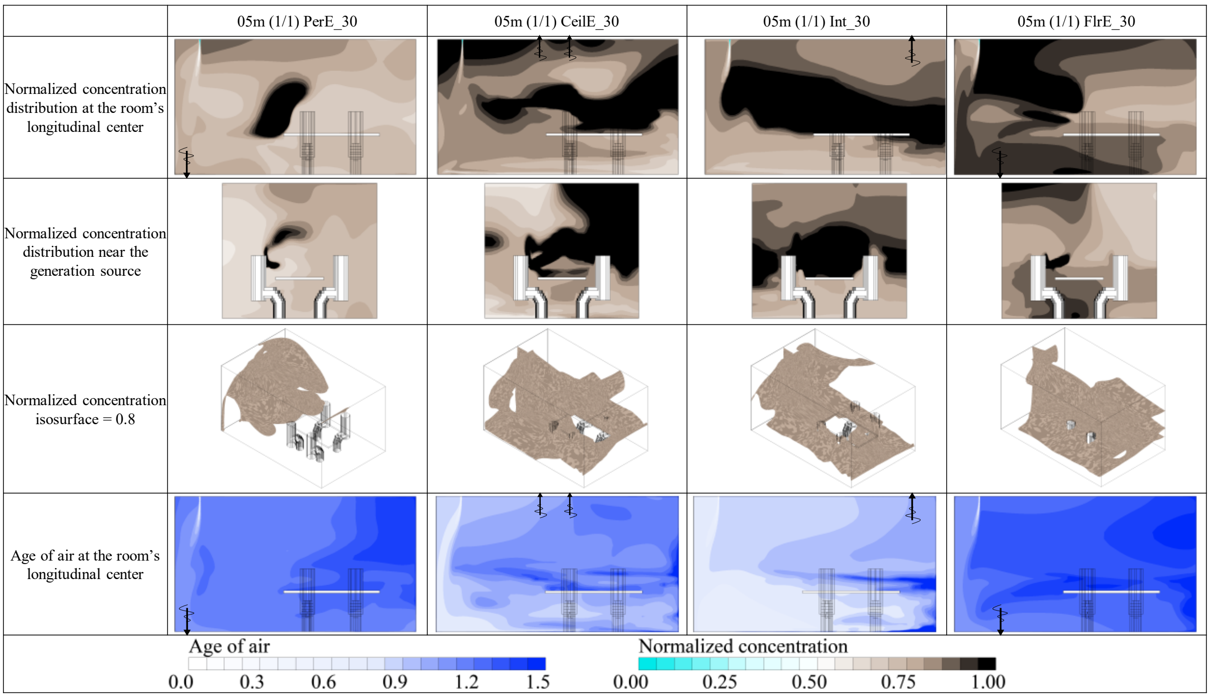


Figure 8 1/1 outlet mode of the slot line diffuser; normalized concentration and age of air's distribution in the analysis space when choosing the different exhaust methods (perimeter floor, ceiling's center, interior's ceiling, and the floor exhaust)

	05m (1/2) PerE_30	05m (1/2) CeilE_30	05m (1/2) Int_30	05m (1/2) FlrE_30
Exhaust temperature (°C)	22.09	23.04	22.11	21.79
AC's capacity (W)	515.44	453.53	513.53	534.73
Average air temperature in the occupied zone (°C)	24.24	22.71	21.59	22.13
Average temperature of glass inner surface (°C)	8.69	8.34	8.50	8.34

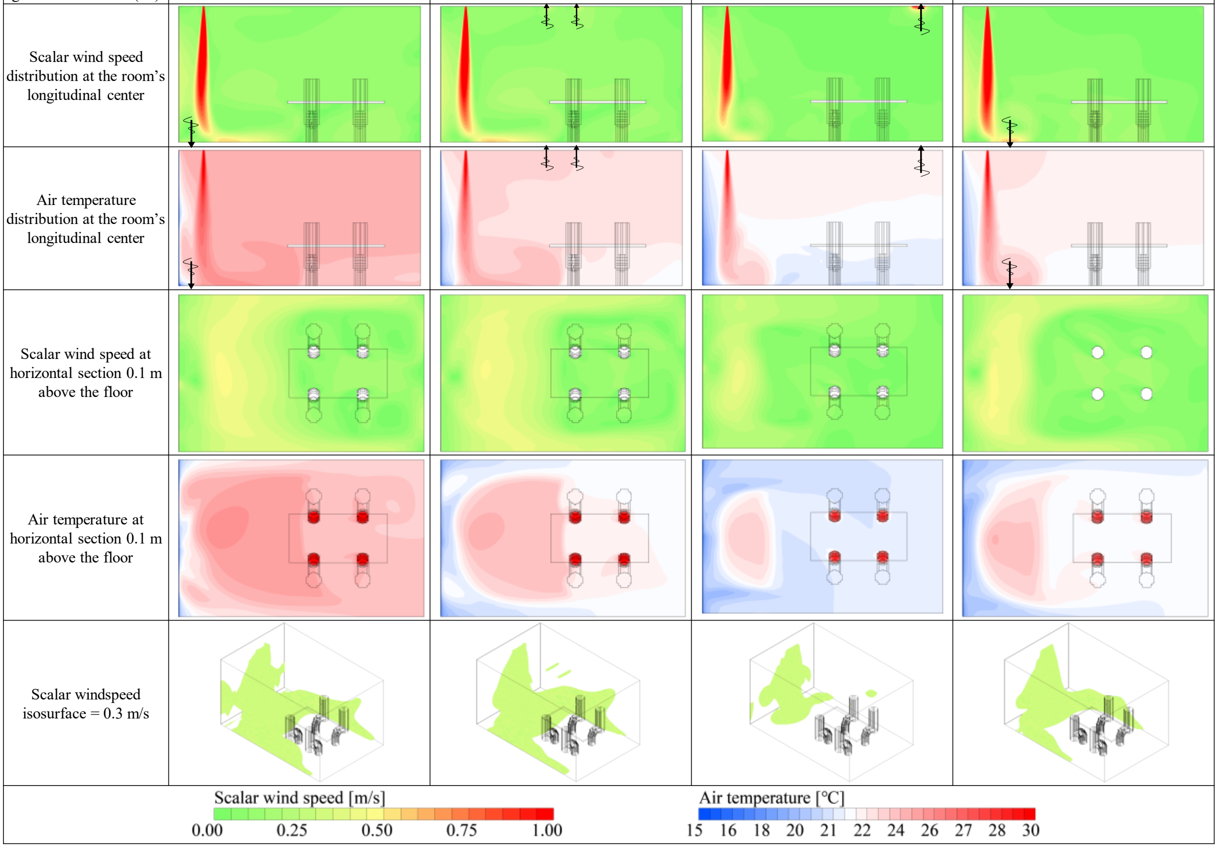


Figure 9 1/2 outlet mode of the slot line diffuser; air temperature and scalar wind speed distribution in the analysis space while using the different exhaust methods (perimeter floor, ceiling's center, interior's ceiling, and the floor exhaust)

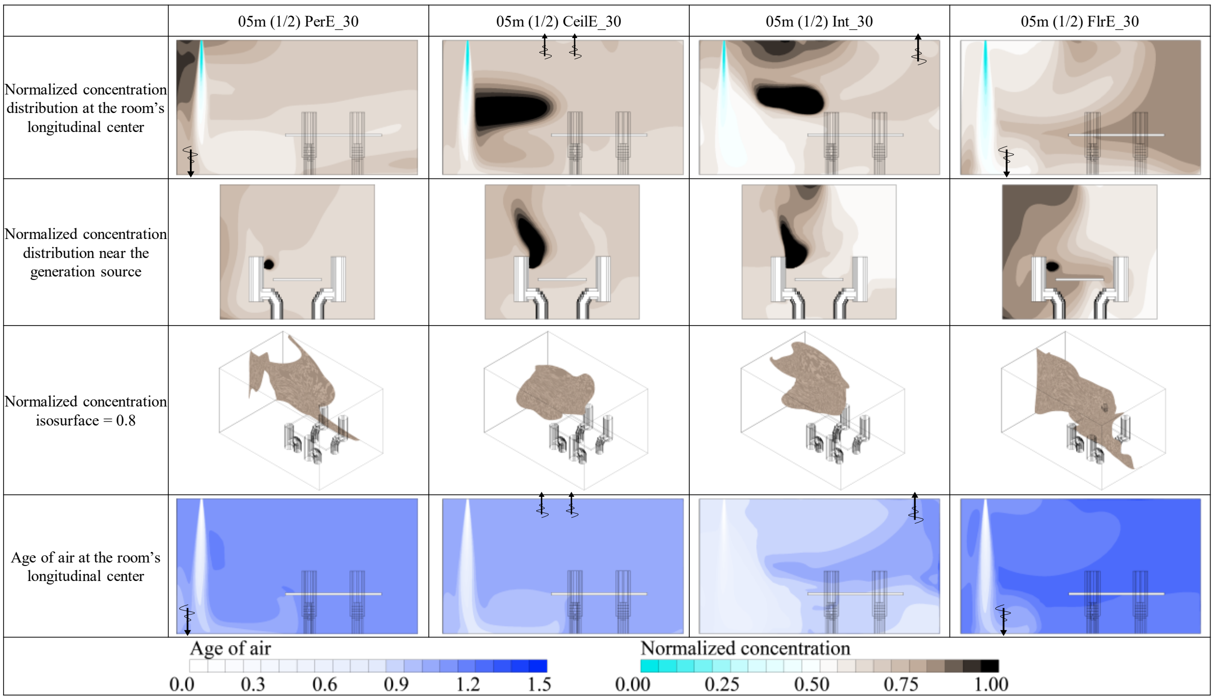


Figure 10 1/2 outlet mode of the slot line diffuser; normalized concentration and age of air's distribution in the analysis space when choosing the different exhaust methods (perimeter floor, ceiling's center, interior's ceiling, and the floor exhaust)

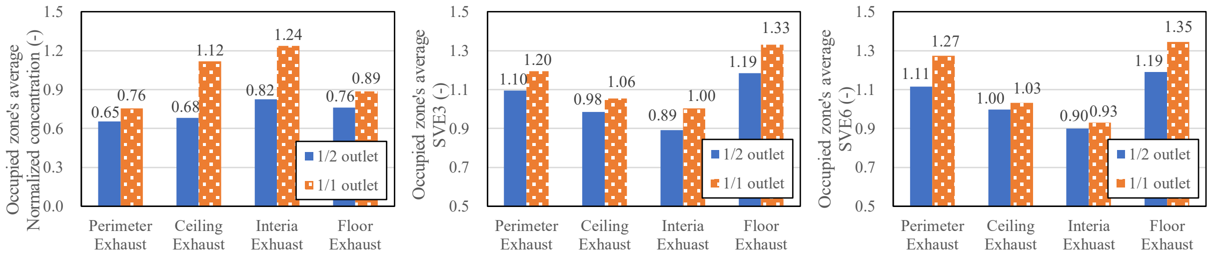
3.2 Ventilation performance

The normalized concentration C^* was used to represent the tracer gas concentration distribution in the analysis space. The normalized concentration C^* can be expressed by ***Function (1)*** below, which C_P means the concentration data of the data pickup point.

$$C^* = \frac{C_P}{C_{EA}} \tag{1}$$

Additionally, two scales for ventilation efficiency, the age of air (SVE3) and the residual lifetime of air (SVE6), are calculated and listed.

In ***Fig. 8*** and ***Fig. 10***, the distribution of C^* is shown by contour graphs, in which the sections are located on the room's longitudinal center and vertical to the generation source (human simulator's face). The isosurface of C^* = 0.8 in 3D view, and the SVE3's distribution in the room's longitudinal center are also shown in these two graphs, which correspond to the diffuser's 1/2 and 1/1 supply modes, respectively. A comparison of the mean volume of C^*, SVE3, and SVE6 in the occupied zone under different supply and exhaust methods shows in ***Fig. 11***.



(a) (b) (c)

Figure 11 The normalized concentration C^*, the age of air (SVE3), and the residual lifetime of air (SVE6)'s relationship with the slot line diffuser's air supply modes and the room's exhaust methods (a) average normalized concentration in the occupied zone; (b) mean age of air (SVE3) in the occupied zone; (c) mean residual life time of air (SVE3) in the occupied zone

4 CONCLUSIONS

The heating and ventilation performance of the slot line diffuser in a typical office space with large area exterior glass is numerically investigated. Two panels are installed inside the diffuser, which is considered to improve its winter performance by converging the supply airflow. Under the diffuser's standard heating supply condition (30℃, 200m3/h, 4.5 ACPH), CFD simulations using the low Reynolds number turbulence model, the detailed air supply characteristics model, and fine meshes were conducted. The diffuser's supply mode (1/1 or 1/2 outlet) and exhaust methods are adjusted to clarify their impact. From the air temperature, airflow, and tracer gas distribution in the occupied zone and the thermal performance and ventilation efficiency index calculation, it can be concluded that:

- Converging the supply airflow by halving the slot line diffuser's supply area can obviously improve this perimeter heating and ventilation performance during the winter.
- Under the diffuser's 1/1 outlet mode, insufficient heating air flow gets into the space near the floor. It can't block cold drafts going into the room's interior without mounting a floor exhaust near the window. A vertical temperature difference of about 10℃ occurred in the occupied zone when using the ceiling or interior exhaust. And it can decrease to around 2℃ while using a floor exhaust.
- Under the 1/2 outlet mode of the diffuser, heated air can overcome buoyance's effect and impinge on the floor. Blocking and heating the cold draft before it enters the room's

interior. It was possible to maintain a vertical temperature difference of less than 2°C in the occupied zone when exhausting from the ceiling of the room and less than 1°C from the floor near the window.

- A flow field similar to the wall confluent jet ventilation is achieved in this office space using a slot line diffuser in 1/2 outlet mode. The mean normalized concentration in the occupied zone is less than 0.7 using the perimeter or ceiling exhaust, and less than 0.82 using other exhaust methods.
- The mean normalized concentration in the occupied zone becomes nearly equal to mixing ventilation (0.76-1.24) when the diffuser supplies the air in the 1/1 outlet mode. However, even though mixing ventilation was considered, the 1/2 outlet mode had a higher SVE performance.
- Though converging the supply air (1/2 supply mode) increased the draft rate (DR) in the interior space, it's still less than 6%, considering less impact on thermal comfort. Meanwhile, it doubled the ADPI index (94%~) compared to 1/1 supply mode, indicating better and satisfactory air distribution performance.
- Combining the ceiling exhaust mounted at the room's center with the slot line diffuser has the best ventilation performance. And mounting a floor exhaust near the exterior glass (between the glass and diffuser's direct below) has the best heating and cold draft-blocking performance.
-

This study validates the ventilation potential of the perimeter slot line diffuser. The best heating performance can be ensured by using a perimeter floor exhaust. And an ideal ventilation and heating performance can be achieved by combining the diffuser's 1/2 outlet with a commonly used ceiling exhaust. Future work will evaluate its ventilation performance under the cooling-supplied mode. It is intended to investigate the possibility of further improving ventilation efficiency by establishing ceiling-supplied displacement ventilation.

5 REFERENCES

[1] Y. Cheng, Y. Cheng, Z. Lin, W. Wu and T. Yao, "Performance evaluation of stratum ventilation with slot diffuser using CFD," in *Indoor Air 2014 - 13th International Conference on Indoor Air Quality and Climate*, Hong Kong, 2014.

[2] G. Cao, P. Nielsen, R. L. Jensen, P. Heiselberg, L. Liu and J. Heikkinen, "Protected zone ventilation and reduced personal exposure to airborne cross-infection," *Indoor Air,* vol. 25, no. 3, pp. 307-319, 2015.

[3] B. Wei and Z. Li, "Simulation of Indoor Air Flow Fields in Buildings With Large Interior Spaces With Stratified Air Conditioning and Perforated Side Wall Diffusers," in *ASME 2010 4th International Conference on Energy Sustainability*, 2010.

[4] F. Lorch and P. Straub, "PERFORMANCE OF OVERHEAD SLOT DIFFUSERS WITH SIMULATED HEATING AND COOLING CONDITIONS," *ASHRAE Transactions,* vol. 89, pp. 200-211, 1983.

[5] W. Rousseau, "Perimeter Air-Diffusion Performance Index Tests for Heating with a Ceiling Slot Diffuser," *ASHRAE Transactions,* vol. 89, pp. 191-199, 1983.

[6] S. Janbakhsh and B. Moshfegh, "Experimental investigation of a ventilation system based on wall confluent jets," *Building and Environment,* vol. 80, pp. 18-31, 2014.

[7] S. Sheng, T. Yamanaka, T. Kobayashi, J. Yuan and M. Katoh, "Modeling of supply airflow from slot line diffuser on ceiling for CFD of thermal environment in perimeter zone," *Building and Environment,* vol. 213, no. 1, 2022.

[8] K. Abe, T. Kondoh and Y. Nagano, "A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows—I. Flow field calculations.," *International journal of heat and mass transfer,* vol. 37, no. 1, pp. 139-151, 1994.

[9] K. Abe, T. Kondoh and Y. Nagano, "A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows—II. Thermal field calculations," *International Journal of Heat and Mass Transfer,* vol. 38, no. 8, pp. 1467-1481, 1995.