



Aalborg Universitet

AALBORG UNIVERSITY  
DENMARK

## Numerical investigation on the impact of different supply air terminal devices on the performance of the newly combined ventilation and heating system

Sadeghian, Parastoo; Polak, Joanna; Afshari, Alireza; Sadrizadeh, Sasan

*Published in:*  
IOP Conference Series: Materials Science and Engineering

*DOI (link to publication from Publisher):*  
[10.1088/1757-899X/609/5/052024](https://doi.org/10.1088/1757-899X/609/5/052024)

*Creative Commons License*  
CC BY 3.0

*Publication date:*  
2019

*Document Version*  
Publisher's PDF, also known as Version of record

[Link to publication from Aalborg University](#)

*Citation for published version (APA):*  
Sadeghian, P., Polak, J., Afshari, A., & Sadrizadeh, S. (2019). Numerical investigation on the impact of different supply air terminal devices on the performance of the newly combined ventilation and heating system. *IOP Conference Series: Materials Science and Engineering*, 609(5), [052024]. <https://doi.org/10.1088/1757-899X/609/5/052024>

### General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- ? Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- ? You may not further distribute the material or use it for any profit-making activity or commercial gain
- ? You may freely distribute the URL identifying the publication in the public portal ?

### Take down policy

If you believe that this document breaches copyright please contact us at [vbn@aub.aau.dk](mailto:vbn@aub.aau.dk) providing details, and we will remove access to the work immediately and investigate your claim.

PAPER • OPEN ACCESS

## Numerical investigation on the impact of different supply air terminal devices on the performance of the newly combined ventilation and heating system

To cite this article: Parastoo Sadeghian *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **609** 052024

View the [article online](#) for updates and enhancements.

# Numerical investigation on the impact of different supply air terminal devices on the performance of the newly combined ventilation and heating system

Parastoo Sadeghian<sup>1\*</sup>, Joanna Polak<sup>2</sup>, Alireza Afshari<sup>2</sup> and Sasan Sadrizadeh<sup>1</sup>

<sup>1</sup> Royal Institute of Technology, Stockholm, Sweden

<sup>2</sup> Aalborg University, Copenhagen, Denmark

\* Parsad@kth.se

**Abstract.** An increased focus on energy saving has led to a rapid development of energy-efficient buildings. In the residential buildings, space heating, ventilation and air conditioning (HVAC) have the highest energy use. The ventilation system is the main tool to provide acceptable indoor air quality and thermal comfort for occupants. This study presents an investigation of the thermal environment in a room served by new developed, combined ventilation and heating system. The focus is on different configurations of the supply air terminal device in the studied system. The main goal is to investigate the influence of different supply air parameters, which in this study are flowrate and temperature, on the airflow behaviour and performance of the mixing ventilation. In this regards, three different supply air conditions with two inlet configurations were considered. This work has been carried out numerically and validated with the laboratory measurements. Computational Fluid Dynamics (CFD) simulation was applied in this study to map the airflow patterns and air temperature distribution. The results showed that decreasing supply air temperature and increasing the flowrate provided a uniform temperature distribution for both inlet configurations. Inlet configuration investigated in case1 has lower vertical temperature differences in comparison with case 2.

**Keywords:** CFD simulation, Supply air terminal device, Combined ventilation and heating system, Supply air parameters

## 1. Introduction

Based on the Energy Performance of Buildings Directive, energy efficiency in buildings requires improvements besides considering indoor air quality and thermal comfort in the indoor environment. The ventilation systems are responsible to provide appropriate indoor air quality for occupants; moreover, indoor air quality has an important impact on the productivity and well-being of individuals [ 1 ]. In the residential and office buildings, space heating, ventilation and air conditioning (HVAC) account for having remarkable energy use up to 35% just in Sweden [ 2 ]. Therefore, energy efficiency, thermal comfort and indoor air quality should be well addressed and enhanced in the development of novel HVAC systems[ 3 ].

One of the current main challenges is increasing energy efficiency and sustainability in existing buildings [ 4 ]. Chen et al. (2015) conducted a numerical study on the effects of supply air parameters and different air supply devices such as mixing supply device (MSD), wall confluent jets supply device (WCJSD), impinging jet supply device (IJSD) and displacement supply device (DSD) on the performance of ventilation system [ 5 ]. Their results showed that WCJSD and IJSD provided a good indoor environment conditions besides being more energy efficient.

Several factors have impact on the performance, design and achievement of ventilation systems such as supply air velocity and temperature, air inlet configurations and type of buildings [ 6 ]. This research study investigated the influence of different supply air terminal devices and supply air parameters on

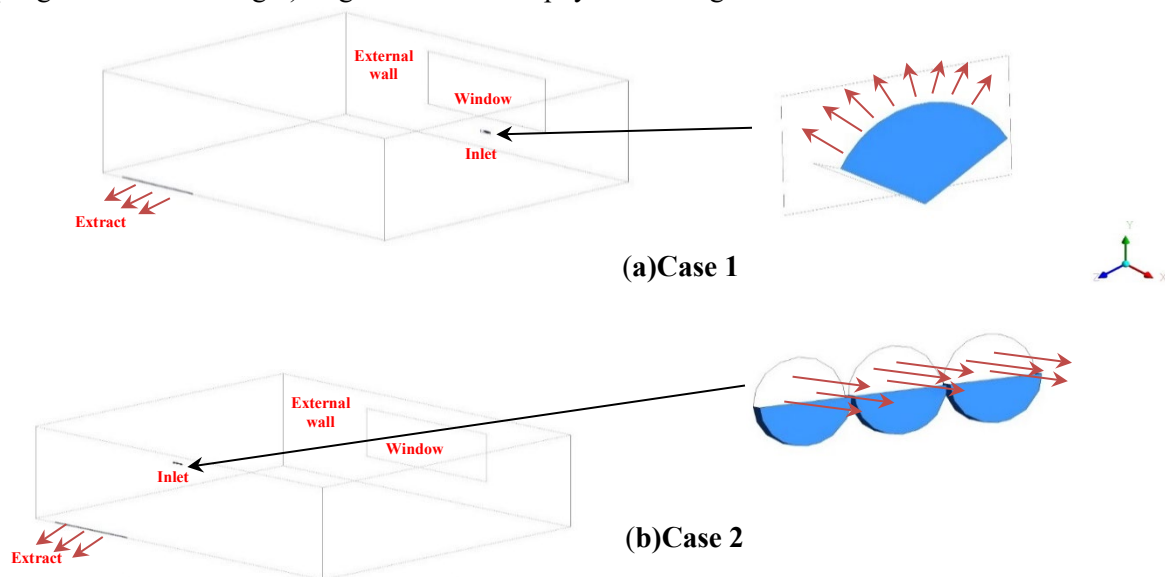


the performance of new combined ventilation and heating system and airflow behaviour in an unoccupied room. Computational Fluid Dynamics (CFD) is applied to quantify the influence of supply air temperature and flowrate on the airflow pattern in the room utilized with mixing ventilation for two different inlet configurations. The results from experimental measurements are used for validation of the simulations.

## 2. Methods

### 2.1 The case study

A room with identical geometry of experimental studied room with two different inlet configurations and positions are selected for simulations in this research. The dimensions are 3.1 m × 3.6 m × 2.7 m (length × Width × Height). Figure 1 shows the physical configuration of the two rooms.



**Figure 1.** Isometric view of the simulated room: (a) case 1 and (b) case 2

The simulated rooms consist one external wall and window with negative heat flux to simulate the cold surface while constant temperature is imposed to ceiling, floor and internal walls. In both cases, the outlet air extracted through a 5mm width exhaust similar to the gap below the door of experiment case. In the case 1, presented in the Figure 1 (a), the supplied air entered the room through the terminal device that is located on the external wall and below the window. In the case 2, the air is supplied to the room through three nozzles located on the upper part of the wall opposite the window as shown in Figure 1 (b). In order to identify the optimum supply air condition for having a satisfying thermal comfort condition for the developed system three different supply air temperatures and airflow rates were simulated for both inlet configurations that the input parameters are summarized in Table 1. The overall heat loss in all the simulated cases are the same as the measurement cases. The three inlet parameters in Table 1 were selected based on the two measurement scenarios that were  $T_{\text{supply air}}=51^{\circ}\text{C}$ ,  $q_{\text{Supply air}} = 7$  (l/s) and  $T_{\text{supply air}}=36^{\circ}\text{C}$ ,  $q_{\text{Supply air}} = 14$  (l/s).

The measurement data collected at supply air temperature of  $51^{\circ}\text{C}$  and flow rate of 7 (l/s) for both configurations and for the inlet configuration 1 the adiabatic boundary condition and for inlet configuration 2 the constant temperature of the  $21^{\circ}\text{C}$  for all the walls were applied, moreover, the constant overall heat loss of -234 W was considered through the window in both inlet configurations. All of the CFD models were validated based on the measurement data, but the details of the validation and measurements methods and data are out of the scope of this conference paper and soon will be published as a journal paper.

**Table 1.** Air supply parameters and boundary conditions of three cases

Case	$T_{\text{supply air}} (^{\circ}\text{C})$	$q_{\text{Supply air flow rate}} (\text{m}^3/\text{s})$	Heat loss through window( $\text{W}/\text{m}^2$ )	Heat loss through external wall( $\text{W}/\text{m}^2$ )	Temperature of the ceiling, floor and internal walls( $^{\circ}\text{C}$ )
<b>a</b>	48	0.006	-72.31	-10.8	21
<b>b</b>	40	0.009	-72.31	-10.8	21
<b>c</b>	32	0.012	-72.31	-10.8	21

## 2.2 Numerical Modeling

In order to simulate the airflow in the case studies, the governing equations of mass, momentum and energy conservation were considered. The turbulent flow was modelled by applying Realizable k- $\epsilon$  model [ 7 ], which is among the common turbulence models for predicting of indoor airflow [ 8 ]. The governing equation can be expressed as

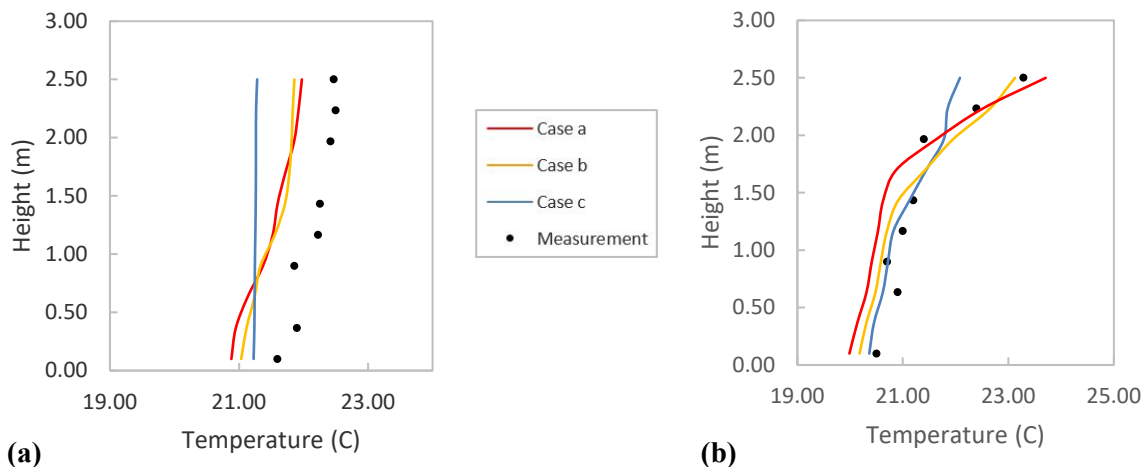
$$\frac{\partial(\rho\phi\vec{V})}{\partial t} + \nabla \cdot (\rho\vec{V}\phi) = \nabla \cdot (\Gamma_{\phi}\nabla\phi) + S_{\phi} \quad (1)$$

Where  $\phi$  represents the transport quantity,  $\vec{V}$  is the air velocity vector,  $\rho$  is the air density,  $\Gamma_{\phi}$  is the effective diffusivity and  $S_{\phi}$  is the source term.

The ICEM CFD is applied to subdivide the geometry into 1.08 million hexahedral cells; moreover, the mesh independency test for coarse (0.57 million cells), medium (1.08 million cells) and fine (1.5million cells) grids showed no significant differences between the fine and medium grid for prediction of airflows. The commercial CFD code ANSYS Fluent 19.2 was applied for all numerical simulations. The numerical models were validated with a laboratory measurement in a full-scale laboratory.

## 3. Results

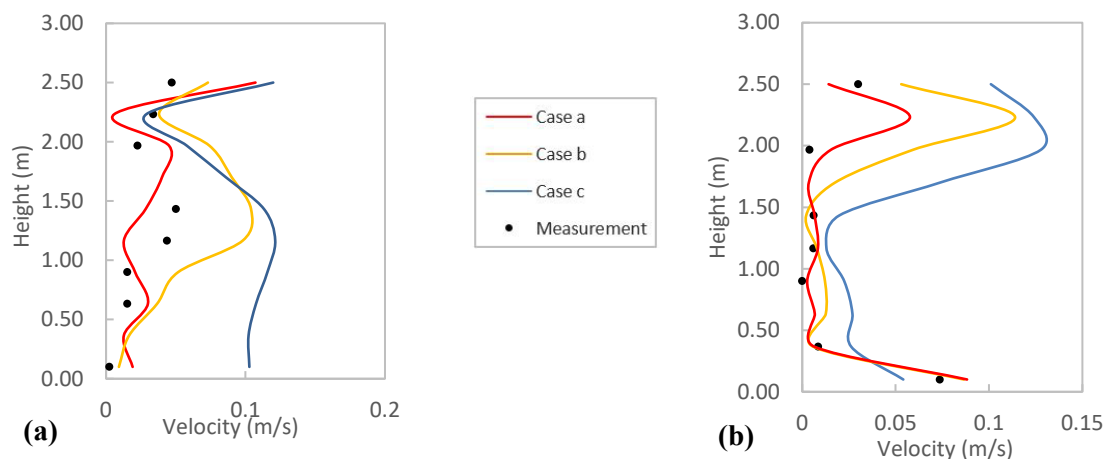
The combination of the two inlet configurations and three supply air cases resulted in six simulation cases. Figure 2 shows the vertical temperature distributions in the middle part of the room at 1.55 m distance from the window for the six simulated cases. Increasing inlet airflow rate with reduction in supply air temperature resulted in a more uniform vertical temperature distribution for both cases based on simulated results.



**Figure 2.** Comparison of vertical temperature distribution for the supply air cases in the middle of the room at 1.55 m distance from the window for (a) Case 1 and (b) Case 2

In figure 3, the vertical velocity distribution in the middle part of the room, which is located 1.55 m distant from the window, is presented for all simulated cases besides measurement data. The velocity distribution shows the impact of supply air velocity on the airflow pattern in the room. In case 1, increasing the supply air velocity, rises the velocity in middle part while in case 2, the increase in air velocity is obtained on upper part of the room. The results of draught risk assessment for case c at 0.1-2.5 m height from floor are available in Table 2 for both inlet configurations. Comparison of the simulated results with measurements data in figures 2-3 showed that the adopted numerical models successfully predicted the airflow velocity and temperature with the same trend as the measurement case.

Figure 4 displays temperature contour plot for the simulated cases at the cross section passing the centre of the room. As presented in Subfigures 4(a-c), the vertical temperature differences for the inlet configuration 1 gradually decreased by reducing supply air temperature and increasing the inlet flowrate from case a ( $T=48^{\circ}\text{C}$  -  $q=6$  L/s) to case c ( $T=32^{\circ}\text{C}$  -  $q=12$  L/s). The subfigures 4(d-f) show that the temperature distribution for inlet configuration 2 gradually became more uniform in occupants' zone by decreasing the supply air temperature and increasing the inlet flowrate from subfigure 4(d) to subfigure 4(f).



**Figure 3.** Comparison of vertical velocity distribution for the supply air cases in the middle of the room at 1.55 m distance from the window for (a) Case 1 and (b) Case 2

**Table 2.** Draught risk assessment for case 1-c and case 2-c (0.1-2.5m height)

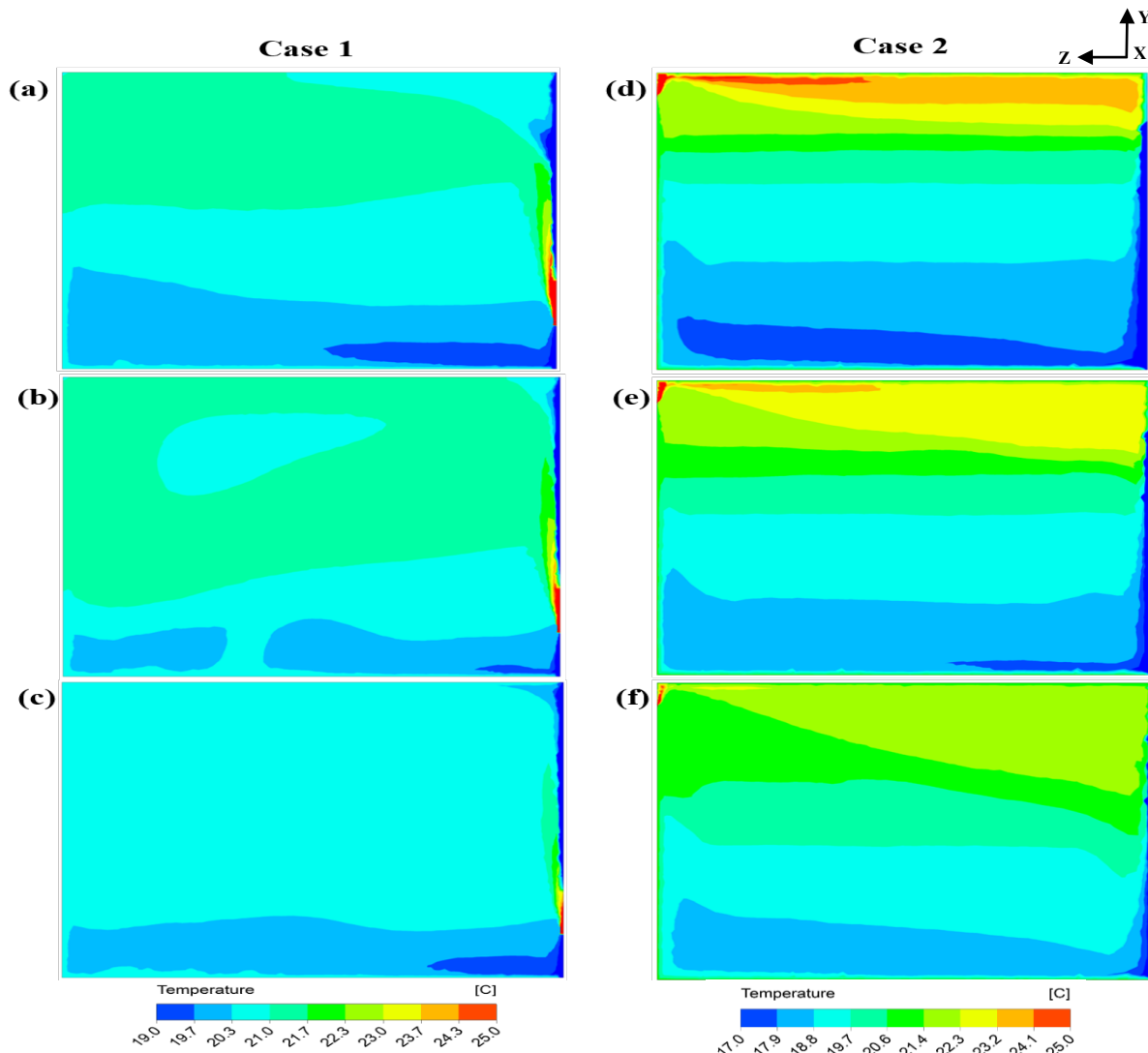
Case	Max velocity(m/s)	Average velocity(m/s)	Average PD due to draught(%)	Max PD due to draught (%)
1-c	0.12	0.1	6.99	11.07
2-c	0.13	0.06	5.41	15.65

#### 4. Discussion

The simulated results and experimental data for two inlet configurations in figure 2 and 3 indicate that the location and configuration of supply air terminal devices have influences on the airflow behaviour and thermal condition of the room. Case 1 with supply air temperature of  $48^{\circ}\text{C}$  and flow rate of 6 (l/s) resulted to less than  $1^{\circ}\text{C}$  vertical temperature differences in occupied zone, however, in the same supply air condition, case 2 showed  $1.7^{\circ}\text{C}$  vertical temperature differences. Consequently, case 1 presented more uniform temperature distribution in comparison with case 2. As shown in Figure 2, the maximum vertical temperature differences for both inlet configurations are less than 3 degree in the occupied zone, which is in the acceptable range for thermal comfort of occupants based on ASHARE standard 55 [ 9 ].

Due to different locations of both supply air terminal devices, the maximum velocity with supply air flowrate of 12 (l/s) were 0.12 (m/s) for case 1 and 0.09 (m/s) for case 2 in the occupied zone. Since case 1 is located under the window, higher air mixing in occupied zone was achieved that created lower vertical temperature differences in contrast with case 2.

The simulated results were also compared with measurement cases in figures 2 and 3. For both configurations, the measurements have higher vertical temperature differences in comparison with CFD results because the supply air temperature for experimental cases were  $51^{\circ}\text{C}$ , but the vertical velocity distributions were smoother in comparison with case b and c due to lower inlet flowrate in measuring cases. Increasing the inlet airflow rate with decreasing the supply air temperature, as presented in figure 2 and 4, reduce the vertical temperature differences and temperature distribution became uniform for both inlet configurations in occupied zone. Consequently, the thermal condition of room will be more satisfactory based on the thermal comfort standards [ 9 ].



**Figure 4.** Temperature contour plot at cross section passing the centre of the rooms(Y-Z plane) for (a) case1 and (d) case2 with  $T_{\text{supply air}}=48^{\circ}\text{C}$ ,  $q_{\text{Supply air flow rate}} = 6$  (l/s); (b) case1 and (e) case2 with  $T_{\text{supply air}}=40^{\circ}\text{C}$ ,  $q_{\text{Supply air flow rate}} = 9$  (l/s); (c) case1 and (f) case2 with  $T_{\text{supply air}}=32^{\circ}\text{C}$ ,  $q_{\text{Supply air flow rate}} = 12$  (l/s)

Considering the presented results in figure 3, increasing supply airflow rate rises the velocity in areas above the inlet and increases the risk of dissatisfaction of occupants due to draught. In this regards, draught risk assessment is essential for case c that has high supply air flowrate. The calculated results in table 2 showed that the average percentage dissatisfactions (PD) due to draught are 6.99 and 5.41 for case 1-c and case 2-c. For both inlet configurations, the PD values are in acceptable range for thermal comfort [ 10 ].

## 5. Conclusions

This study experimentally and numerically investigated the impact of supply air inlet configurations and supply air parameters on the indoor airflow behaviour and the performance of newly combined ventilation and heating system in an unoccupied room. Two inlet configurations with different locations investigated: circular valve located under the window, case 1 and three nozzles located upper part of the wall opposite the window, case 2. For each inlet configurations, the influence of increasing supply air flowrate and decreasing temperature on vertical temperature and velocity distribution were investigated. The obtained simulated and measurement results showed that both configurations can be applied for supply air inlet for heating the room.

The inlet configuration for case 1, showed reduction in vertical temperature difference to less than 0.5 °C and increase in the vertical velocity distribution at higher supply air flowrates in the occupied zone. Moreover, decreasing the supply air temperature resulted to more uniform temperature distribution in the simulated room. Inlet configuration in case 2, had higher vertical temperature differences across the room height about 1.7°C for case a and resulted in higher velocities at upper part of the room about 0.1 (m/s) for case c. Since the supply air inlet in case 2 is located on the wall opposite the cold surface, less mixing effect was observed compared with case 1.

Based on the achieved results, the location and inlet configuration of case 1 is recommended for providing a satisfied thermal condition for occupants in the indoor environments. Increasing the inlet flowrate and decreasing the temperature of supply air for both inlet configurations showed a drop in the vertical temperature differences and improvement on the performance of mixing ventilation system.

## References

- [ 1 ] Fisk WJ, Rosenfeld AH, 1997, Estimates of improved productivity and health from better indoor environments, *J. Indoor Air* **7**:158e72.
- [ 2 ] Sayigh A, 2013, Sustainability, Energy and architecture: case studies in realizing green buildings. *Elsevier Science*.
- [ 3 ] Noh, K, Han C and Oh M, 2008, Effect of the airflow rate of a ceiling type air-conditioner on ventilation effectiveness in a lecture room, *International Journal of Refrigeration*, **31** 180–188.
- [ 4 ] Cao G, Awbi H, Yao R, Fan Y, Sirén K, Kosonen R and Jensen J, 2014, A review of the performance of different ventilation and air flow distribution systems in buildings, *J. Building and Environment*, **73** 171–186.
- [ 5 ] Chen H, Janbakhs S, Larsson U, and Moshfegh B, 2015, Numerical investigation of ventilation performance of different air supply devices in an office environment, *J. Building and Environment*, **90** 37–50.
- [ 6 ] Chow Æ, Tsang T, Fong C, and Chan L, 2005, CFD study on effect of the air supply location on the performance of the displacement ventilation system, *J. Building and Environment*, **40**, 1051–1067.
- [ 7 ] Air I, Munksgaard B, Issn IAIR, 2003, Quality control of computational fluid dynamics in indoor environments. *J. indoor air*, **13** 2–17.
- [ 8 ] Peng L, Nielsen P V, Wang X, Sadrizadeh S, Liu L, Li Y, 2016, Possible user-dependent CFD predictions of transitional flow in building ventilation. *Build. Environ.* , **99** 130–141
- [ 9 ] ASHRAE. ANSI/ASHRAE standard 55-1992, 1992, Thermal environmental conditions for human occupancy, Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc. Meddelande M85: 24.
- [ 10 ] Fanger P, Melikove A, Hanzawa H and Ring J, 1988, Air turbulence and sensation of draught, *J. Energy and Building* **12** 21-39