DOI : https://doi.org/10.56452/2023-04/003

ISSN: 0974-5823 Vol. 8 No. 4 April, 2023

International Journal of Mechanical Engineering

Impact of Using a Novel Air-Outlet Geometry on the Thermal Comfort Conditions

Ahmed Awwad[1,*], Abouelmagd Abdelsamie[1], Mohamed H. Mohamed[1,2], M. Fatouh[1]

[1]Mechanical Power Engineering Dept., Faculty of Engineering at El-Mattaria, Helwan University, P.O. 11718, Cairo, Egypt
[2]Mechanical Engineering Dept., Collage of Engineering and Islamic Architecture, Umm Al-Qura University, P.O. 5555, Makkah, Saudi-Arabia

*Corresponding author. E-mail address: Ahmedawwad@m-eng.helwan.edu.eg

Abstract

Square louver face ceiling diffuser is well known outlet in the air conditioning applications. Small change in the geometry of this diffuser causes changes in the air distribution and air quality inside the room. With the aid of the computational fluid dynamics, different simulations are performed to obtain the optimum design of the diffuser by changing the blade and lip angles of the standard diffuser using room dimensions of 2.75 x 2.75 x 2.7 m. The results showed that the diffuser shape with blade angle of 65º and lip angle of 0º (Case-I) in the considered room reaches the comfort conditions inside the room (i.e., temperature difference of 1.64 ºC and terminal velocity below 0.25 m/s and). In addition, the diffuser model with a blade angle of 45º and a lip angle of 0º (Case-II) is suitable for high-height spaces with a terminal velocity of 0.20 m/s and a temperature difference of 1.65 ºC. Using a blade angle of 60º and a lip angle of 15º, (Case-III), the air terminal velocity is about 0.20 m/s, and the air temperature difference is 1.24 ºC.

Keywords: Ceiling diffusers, Comfort criteria, Blade angle, Lip angle, CFD.

1. Introduction

The main goal of an air distribution system is to provide thermal comfort to the human occupants. The primary factor in determining the thermal comfort condition of a room is the incoming supply airflow, so considerable attention is paid to the exit airflow from various types of supply air terminal devices/outlets. The ASHRAE Handbook of fundamentals [1] explains and discusses air distribution for air terminal equipment in rooms, which represents a major factor in achieving comfortable conditions for space occupants and minimizing energy consumption. Among them, primary air provides the driving force for indoor air movement. The most important step in achieving efficient comfort regulation is therefore the selection of the optimum supply air opening, such as a louvred ceiling diffuser. Mohamed [2] studied the distributions of temperature and air velocity inside the room for simple and non-simple ceiling diffusers (round and square) using experimental and numerical techniques. The results showed that, the velocity and temperature distributions from experimental and CFD for simple model are agree very well with non-simple models. Sun and

DOI : https://doi.org/10.56452/2023-04/003

Smith [3] explained the effect of diffuser lip and offset in air flow pattern and temperature distributions inside the room numerically and experimentally. The results presented that, the diffuser lip reduces the total effective flow area which results in an increase the velocity magnitude and produces airflow pattern attach with the ceiling which minimize draft phenomena and enhance surface effect. Aziz et al. [4] investigated numerically and experimentally the performance of three ceiling diffusers (with standard geometry) in indoor airflow pattern to predicate the thermal comfort region with study energy consumption. The maximum energy consummation value recorded from the square shape, while the minimum values belong to the vortex one. Djunaedy and Cheong [5] Shown the airflow throw pattern of the different diffuser simple models numerically (CFD) and compare with the experimental results by Tavakkol et al. [6] (velocity decay coefficient). As recorded, the velocity of the air jet from the simple models begins to decrease faster than the measured velocity profile. Abanto et al. [7] investigated numerically the effect of objectives distribution, supply air inlets and return air outlets positions inside the room on airflow patterns. The results proved that, the values for air temperature, relative humidity, and airflow velocity temperature inside the room are matched with standard recommendation values. Fontanini et al. [8] produced new shape of air outlets based on ductwork fabrication and compared the results numerically with conventional air ceiling diffusers to improve the energy saving inside the buildings (T, V, and power). The results showed that, the room was heated using new fabric ductwork diffuser faster than in case the conventional ceiling diffuser with high performance.

Posner et al. [9] investigated the effect of obstruction walls inside the room on the airflow patterns in both experimentally and numerically methods. It is shown that, the obstruction walls have a strong effect on airflow distribution and must take in consideration at ventilation design system. Tina et al. [10] explained the effect of contaminant particles concentration inside the room using CFD technique and comparing the results with the experimental data reported by Posner [9]. In this study, many turbulence models are used and shown good agreement with the experimental results. Also, Tina et al. [11] studied numerically the behaviours of airflow patterns throughout two room models with and without contaminant particles. In the second room model, the concentrations of contaminant particles from LES are lower than the experimental measured data from Weizhen Lu et al. [12].

Fan et al. [13] numerically and experimentally study the performance of new design ventilation system (diffuse ceiling ventilation concept) in a typical office room, and determine the airflow speed and temperature characteristics at various air change rate. Both the CFD simulation and experiment results are observed the comfort criteria. Zhang et al. [14] used the numerical and experimental methods to investigate the effect of using various air outlets (in standard geometry) with displacement ventilation system to improve the air quality distribution and minimize energy. The results of velocity and temperature from numerical studies are matched and agreed with experimental way. Bin and Sekhar [15] presented new ventilation system and compare the results with the conventional mixing ventilation one experimentally and numerically. The results from this study showed that, the percentage of local fresh air inside the occupied zone was enhanced from the new ventilation system. Liu and Novoselac [16] calculated and studied the air diffusion performance index (ADPI) in heating mode experimentally using various diffuser shapes (square, ceiling linear slot, sidewall grille). The results are agreed with standards ASHRAE standard 70 [17] and with experimental data [18]. Mihailo et al. [19] applied both numerical (CFD) and experimental methods to show the effect of different element configurations inside the plenum box of square ceiling diffuser shape. The results showed that, lower air velocity in flow filed from the case of inclined plate with air guidance compare to others cases, in addition acceptable pressure drop relatively the same value in the cases without equalizing element and inclined perforated plate (uniform permeability).

Awwad et al. [20] studied the effect of change the diffuser blade & lip angles (several new diffuser configurations) numerically on thermal comfort conditions (velocity & temperature) inside the small room (2 m x 2 m x 2m), and compare the results with conventional diffuser. The results observed that, the new configurations reached to the thermal comfort conditions in some cases higher than the conventional one inside the occupied zone due to small room especially the two models (65° blade angle & 0° lip angle) & (60° blade angle & 15° lip angle). On the other hand, the diffuser configuration (45° blade angle & 0° lip angle) results showed that temperature & velocity lower than the conventional one due to the airflow configuration out from the diffuser.

The conclusions of this survey reveal the goals and novelties of the current study. The previous work revealed the following points;

(1) All previous papers present results for conventional air ceiling diffuser configurations, or provide slight modifications,

(2) Propose new ventilation systems and compare the results with those of traditional diffusers,

(3) All previous studies did not consider the effect of louvred ceiling diffuser blades and lip angles on human thermal comfort, and

(4) The performance of new louvred ceiling diffuser configuration in standard sized rooms has not been previously studied or considered.

Referring to the International Building Code (IBC); Article 1208 - Dimensions of Interior Spaces [21], in general, the ceiling height of habitable spaces should not be lower than 2.286 m, with a minimum room size of 6.5 m2. Therefore, the present work aims to examine the thermal comfort conditions of new diffuser configurations in a room size complying with minimum standard requirements. The current simulation results of temperature and velocity distributions for three new louvred ceiling diffusers are compared with those of previous work reported by Awwad et al. [20] to achieve thermal comfort conditions.

This manuscript is organized as follows, Section 1 presents the Introduction, while Section 2 presents the numerical methods and numerical domains. Section 3 reports the simulation results, while Section 4 includes the summary and conclusions of the current work.

2. Methodology

This section presents the numerical models, the governing equations and the employed geometry used in the current simulations. Furthermore, the simulation process (steps) is explained based on previous work reported by Awwad et al. [20].

2.1. Numerical models and governing equations

DOI : https://doi.org/10.56452/2023-04/003

In this study, the ANSYS FLUENT [22] was used to predict the airflow distributions (velocity and temperature) in the room using various shapes of louvred face ceiling diffuser. The standard conservation equations for steady-state incompressible flows (mass, Navier-Stokes, and energy) as mentioned below (Eq. (1), Eq. (2). And Eq. (3) respectively) are solved in conjunction with an achievable k-ε turbulence model.

$$\boldsymbol{\nabla}.\mathbf{v} = 0, \tag{1}$$

$$\boldsymbol{\rho}(\mathbf{v} \cdot \boldsymbol{\nabla})\mathbf{v} = -\boldsymbol{\nabla P} + \boldsymbol{\mu}\boldsymbol{\nabla}^2\mathbf{v} + \boldsymbol{\rho g}, \tag{2}$$

Where v, ρ, P, μ, and g, are the velocity vector, the density, the hydrodynamics pressure, the dynamic viscosity, and the acceleration of gravity vector, respectively.

$$\boldsymbol{\rho C_p V \cdot \nabla T = \lambda_{eff} \nabla^2 T}. \tag{3}$$

Where C_p and λ_{eff} are the specific heat at constant pressure and the effective thermal conductivity which can be described as,

$$\boldsymbol{\lambda_{eff} = \lambda_l + \lambda_t}. \tag{4}$$

Where, λ_l, $\lambda_t$are the laminar and the turbulent thermal conductivity, respectively.

2.2 Geometry and the computational domain

The computational fluid domain is shown in Fig. 1. This figure shows a room with dimensions 2.75 m long, 2.75 m wide, and 2.7 m high. Even though, the supply and exit ducts in the current work still have the same dimensions similar to the work of Awwad et al. [20], the main difference between the current work and that of Awwad et al. [20] is the room size, they tested a room with dimensions 2 m long, 2 m wide and 2 m high. The airflow returns through a single return duct close to the floor (0.1 m above the room floor); it has the same size supply duct. In the present work, three cases will be investigated in the new larger room size:

Case I: A diffuser with a blade angle (β) of 65º and a lip angle (α) of 0º.

Case II: A diffuser with a blade angle (β) of 45º and a lip angle (α) of 0º.

Case III: A diffuser with a blade angle (β) of 60º and a lip angle (α) of 15º.

The results will be divided into two parts, each examining a different point: 1) the behavior of the diffuser in different sized rooms, and (2) the performance of different diffuser models.

DOI : https://doi.org/10.56452/2023-04/003

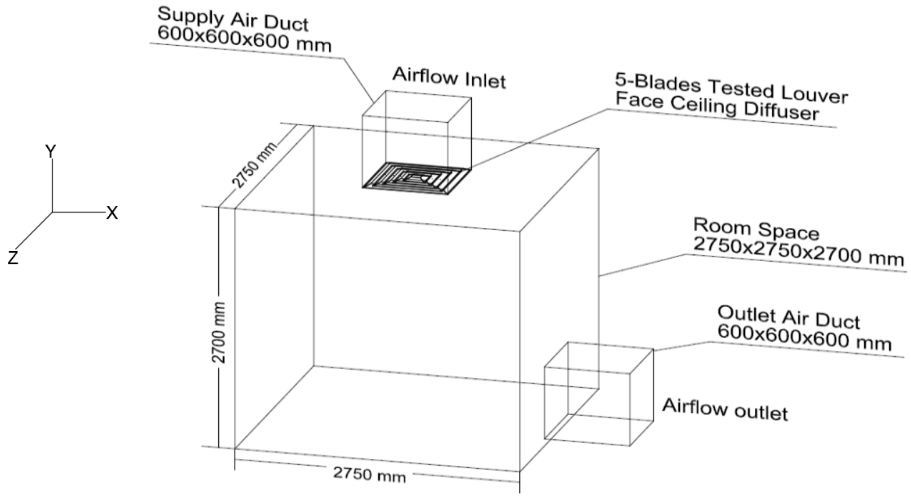


Fig. 1. 3D computational domain

2.3 Simulation Process (Steps)

In the present work and previous work reported by Awwad et al [20], Fig. 2 shows the flow chart of the CFD procedure, the choice of mesh details is selected depends on the geometric model to achieve acceptable mesh quality. The FLUENT setup process is prepared and the realizable k-ε turbulent model is used. The boundary conditions of each region are set in the present work and Awwad et al. [20] with inlet air temperature 287 K and inlet air velocity 0.6 m/s, in addition the walls set with constant temperature 296 K. Finally, the calculation completes when the residual monitor values for all parameters are less than 10^{-3}, except for energy, which is less than 10^{-6}.

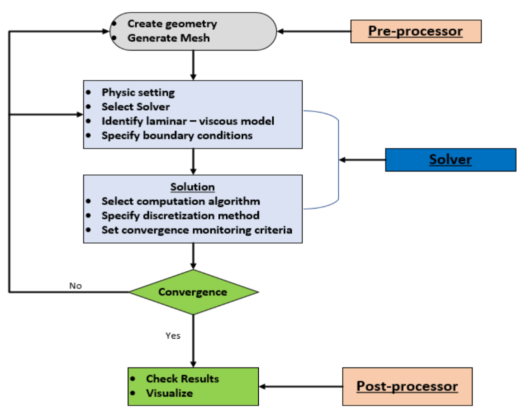


Fig. 2. CFD / numerical study procedures

3. Results and Discussion

In this section, the three cases stated above are investigated in detail. There are two main parameters to evaluate these situations, terminal velocity, and static temperature distribution in the desired space. The results are presented, in the middle of the domain, in three different systematic ways: (1) 2D-cuts as a color contour, and (2) a horizontal line cut at different heights inside the standard requirements for the occupied zone (including a person in a standing or sitting position) as shown in Fig. 3, and (3) the average value of the corresponding line cuts.

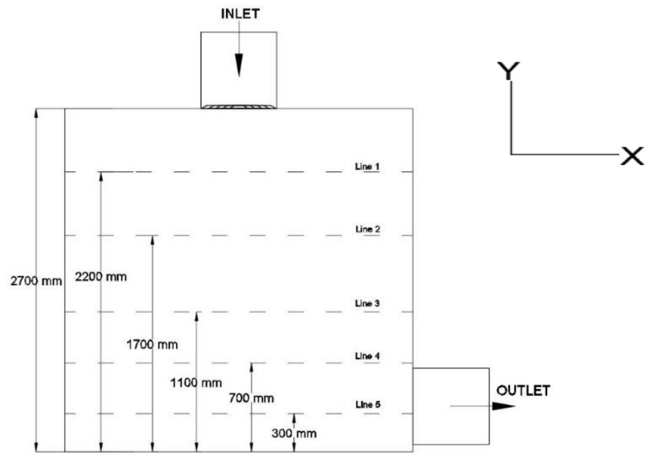


Fig. 3. Line cuts, at different height, through the center line of the domain

3.1 Air velocity and temperature contours of new diffuser configurations

Figure 4 shows the 2D air velocity field in the x-y plane at the centre of the domain for case I. It is clear that the jet velocity at the diffuser is in the range of 4.974 – 5.527 m/s, while the terminal velocity in the room is between 0 – 0.553 m/s. Figure 5 illustrates the air velocity contours for Case II, the displayed jet velocity range is between 2.162 – 2.402 m/s and the terminal velocity range is between 0 – 0.240 m/s. Due to the conical airflow distribution, the terminal velocity is higher near the wall, 0.240 – 0.480 m/s. Figure 6 presents the 2D velocity contours for Case III. The jet velocity is in the range of 3.577 – 3.975 m/s, decreasing with distance from the diffuser channel, and the air distribution velocity in the whole room is in the range of 0 – 0.397 m/s. The flow becomes less attach in case III compared to case I by increasing the lip angle from 0º to 15º.

Table 1 illustrate the air velocity magnitude in current work (Case I, Case II, and Case III), where the velocity values at different line cuts (at 2.2 m, 1.7 m, 1.1 m, 0.7 m, and 0.3 m) corresponding to that in Fig. 3. Table 1 lists the average velocity and temperature for each line cut in Fig. 3. In summary, for Case I, the average value of the terminal velocity is in the range of 0.162 – 0.220 m/s [Lines 1-4], that matched and complied with the thermal comfort standards that recommended by ASHRAE standards 55 [23] (max. 0.25 m/s in occupied areas). However, the average velocity value of the 5th line is close to 0.12 m/s, which is larger than the recommended minimum value (0.1 m/s) in the SMACNA standard [24]. Table 1 presents the air velocity magnitude for Case II, where the average terminal velocity values are shown as 0.15 – 0.17 m/s [lines 1-4] and the velocities near the wall are displayed at high values, which is also in line with the recommendations of ASHRAE Standard 55. However, the average velocity value of Line 5 is about 0.1 m/s, which meets the SMACNA standard [24]. Hence, it is recommended to use the diffuser model (Case II) in rooms with higher heights. In addition, from Table 1, the average velocity values for Case III at different lines is about 0.15 – 0.20 m/s [Lines 1-4], and near the floor at line 5 shown 0.11 m/s. These values also comply with the recommendations of ASHRAE Standard 55 [23] and SMACNA Standard [24]. Also, the values of Case III are close to those of Case I, so it is recommended to use Case III in larger rooms to achieve thermal comfort more quickly and minimize the number of diffusers and ductwork installation.

DOI : https://doi.org/10.56452/2023-04/003

Figures 7, 8, and 9 are the two-dimensional static temperature distribution contours of the x-y plane in the center of the domains of Case I, Case II, and Case III, respectively. Figure 7 shows the static temperature distribution for Case I in the range of 293.3 – 294.2K. Figure 8 shows the 2D temperature distribution of Case II. The temperature distribution in the upper part of the room ranges from 292.398 – 293.298 K. However, in the lower part of the room (near the room floor), the temperature values are in the range of 293.298 – 294.199 K, due to the minimum flow on the ceiling. The temperature distribution of Case III is shown in Fig. 9. Clearly, the temperature range of the upper room side is 292.4 – 293.3 K, while that of the lower room is 293.3 – 294.2 K, which is similar to the temperature distribution of Case 1. However, the temperature distribution on the lower side of the room (near the floor) is almost 293.294 – 294.196 K.

Table 1 tabulates average of velocity and temperature along each line cuts shown in Fig. 3. For case I, the vertical temperature difference between the lines is 1. 64°C. It should be noted that according to ASHRAE Standard 55 [23], the vertical temperature difference within the occupied area must not exceed 3°C.

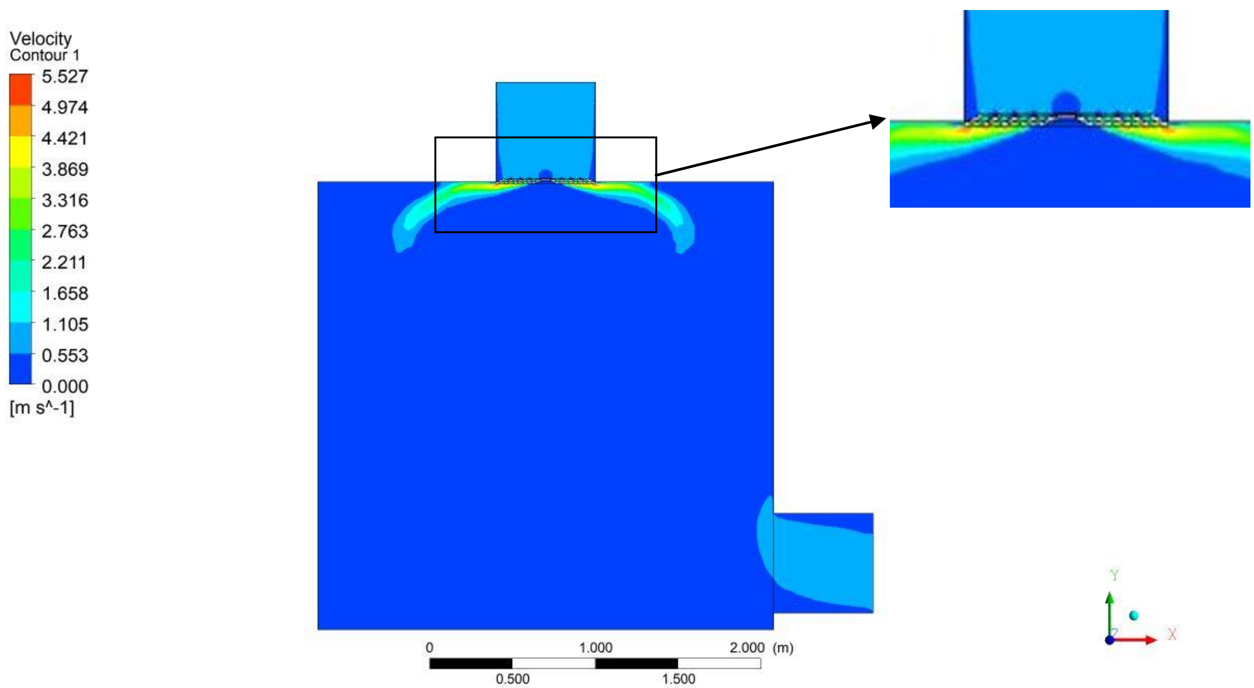


Fig. 4. Velocity magnitude of air flow at x-y plane (2D cut) in the center of the domain, Case I

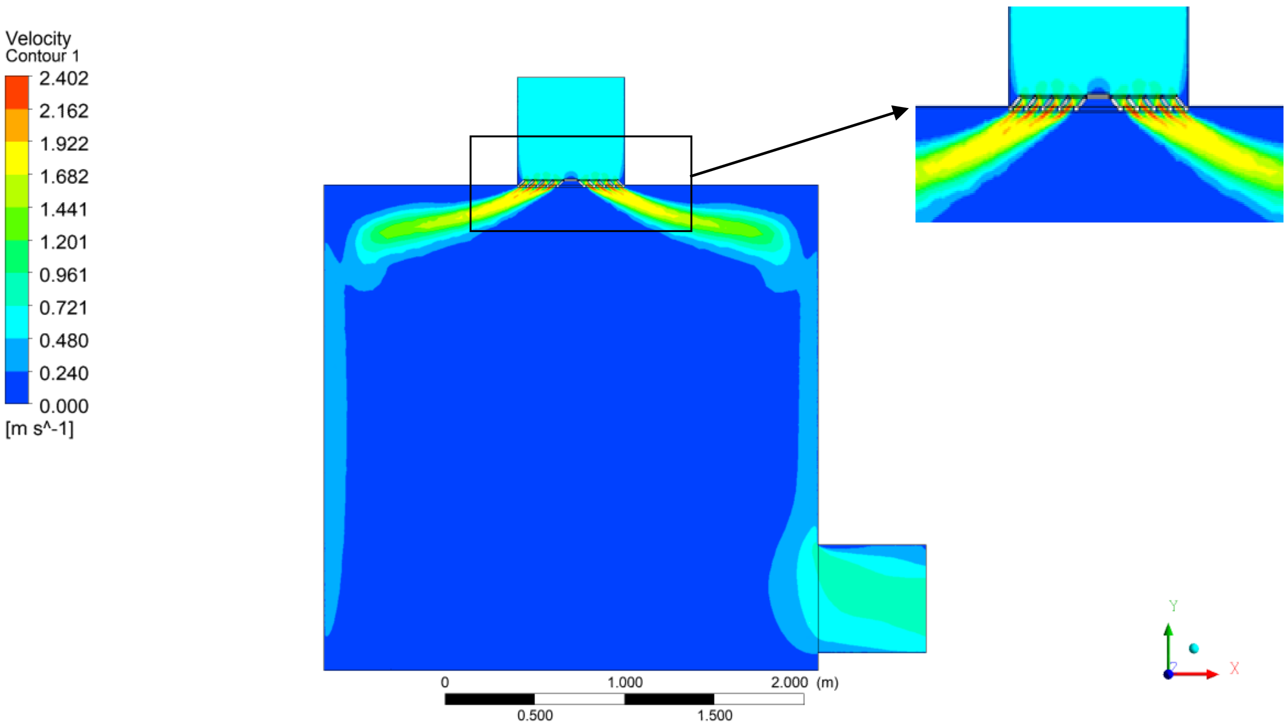


Fig. 5. Velocity magnitude of air flow at x-y plane (2D cut) in the center of the domain, Case II

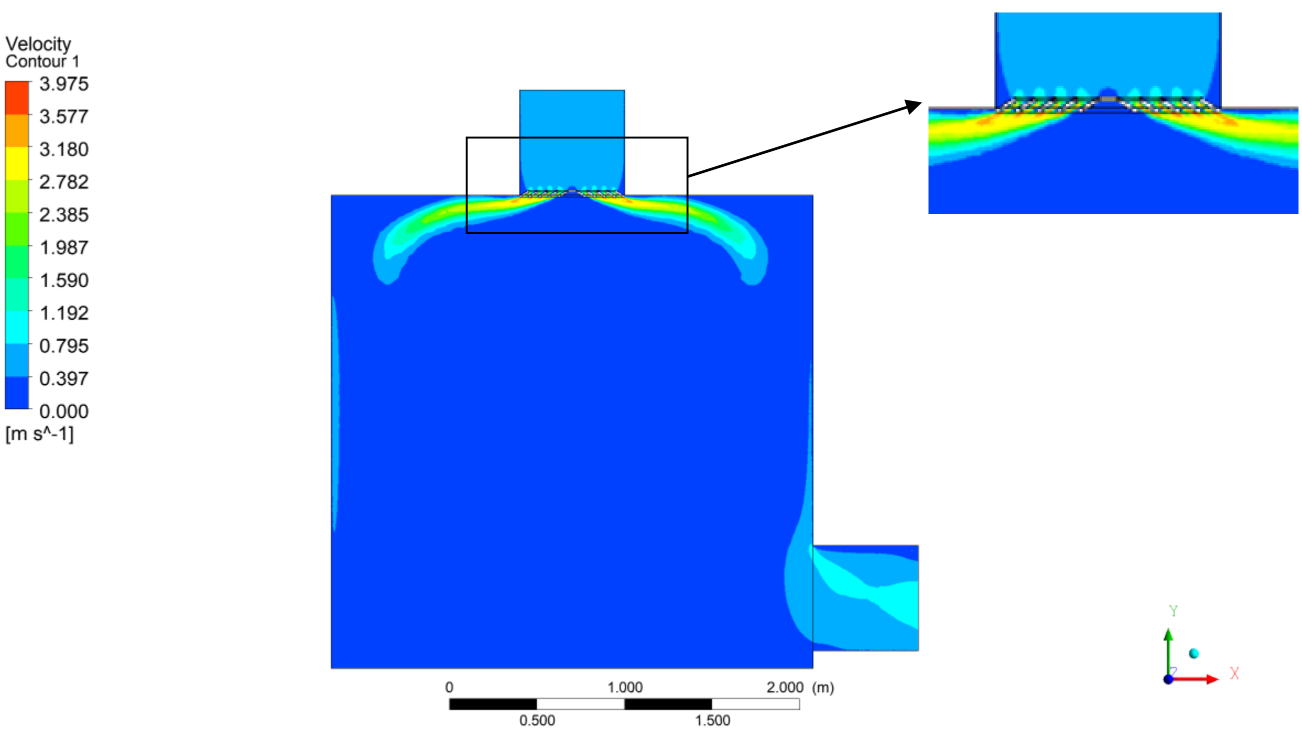


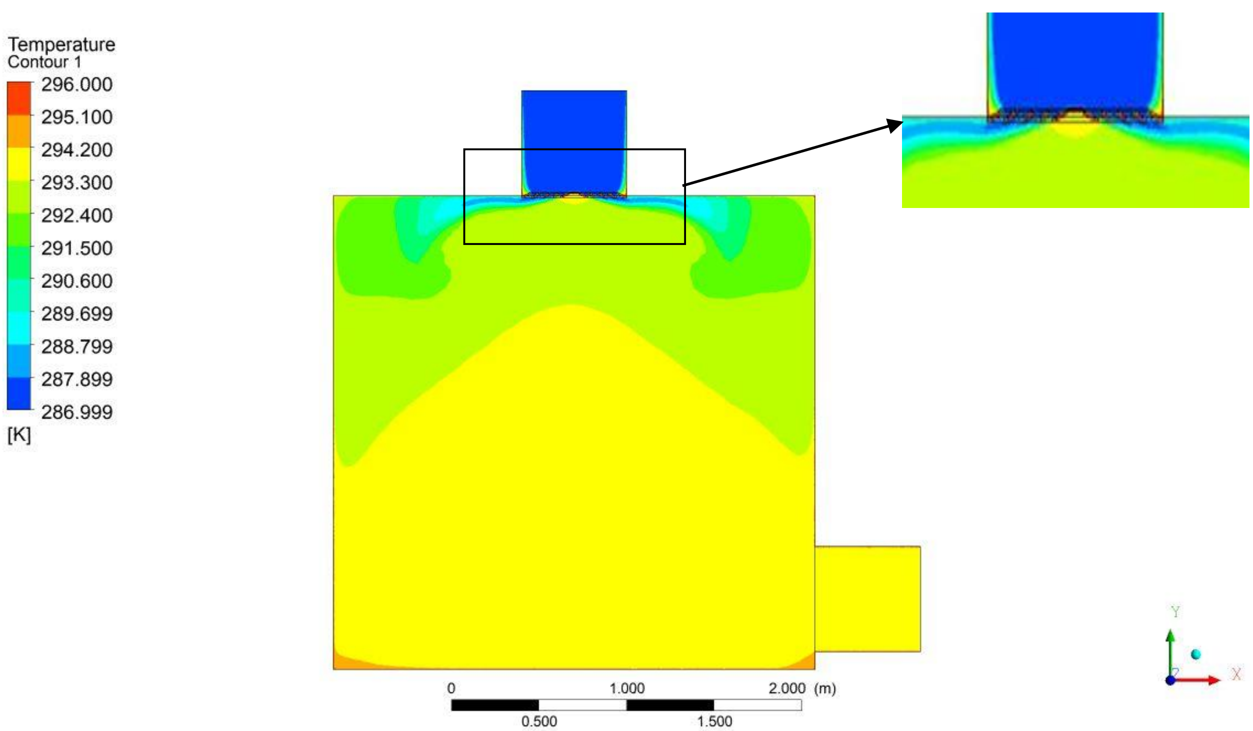
Fig. 6. Velocity magnitude of air flow at x-y plane (2D cut) in the center of the domain, Case III

DOI : https://doi.org/10.56452/2023-04/003

Table 1 Average of air velocity and static temperature along each line cuts presented in Fig. 3, for Case I, Case II, Case III

Line no.	Diffuser configuration					
	Case I (65º blade & 0º lip)		Case II (45º blade & 0º lip)		Case III (60º blade & 15º lip)	
	Average air velocity magnitude (m/s)	Average air static temperature (K)	Average air velocity magnitude (m/s)	Average air static temperature (K)	Average air velocity magnitude (m/s)	Average air static temperature (K)
Line 1	0.220	292.270	0.169	292.100	0.204	292.580
Line 2	0.172	293.270	0.155	292.800	0.161	293.130
Line 3	0.170	293.600	0.153	293.300	0.159	293.540
Line 4	0.162	293.740	0.150	293.500	0.152	293.730
Line 5	0.119	293.910	0.106	293.750	0.113	293.820

Table 1 show the average static temperature at different levels in the investigation room for Case II, where the vertical temperature difference inside the room is about 1.65°C. For Case III, Table 1 provide the average values of the temperature at several locations in the studied room. The temperature difference between the vertical lines is 1.24°C. For Case III and Case II, this diffuser configuration exhibits values of vertical temperature difference within the occupied area that are smaller than those recommended by ASHRAE Standard 55 [23].



DOI : https://doi.org/10.56452/2023-04/003

Fig. 7. Static temperature of the air flow at x-y plane (2D cut) in the center of the domain, Case I.

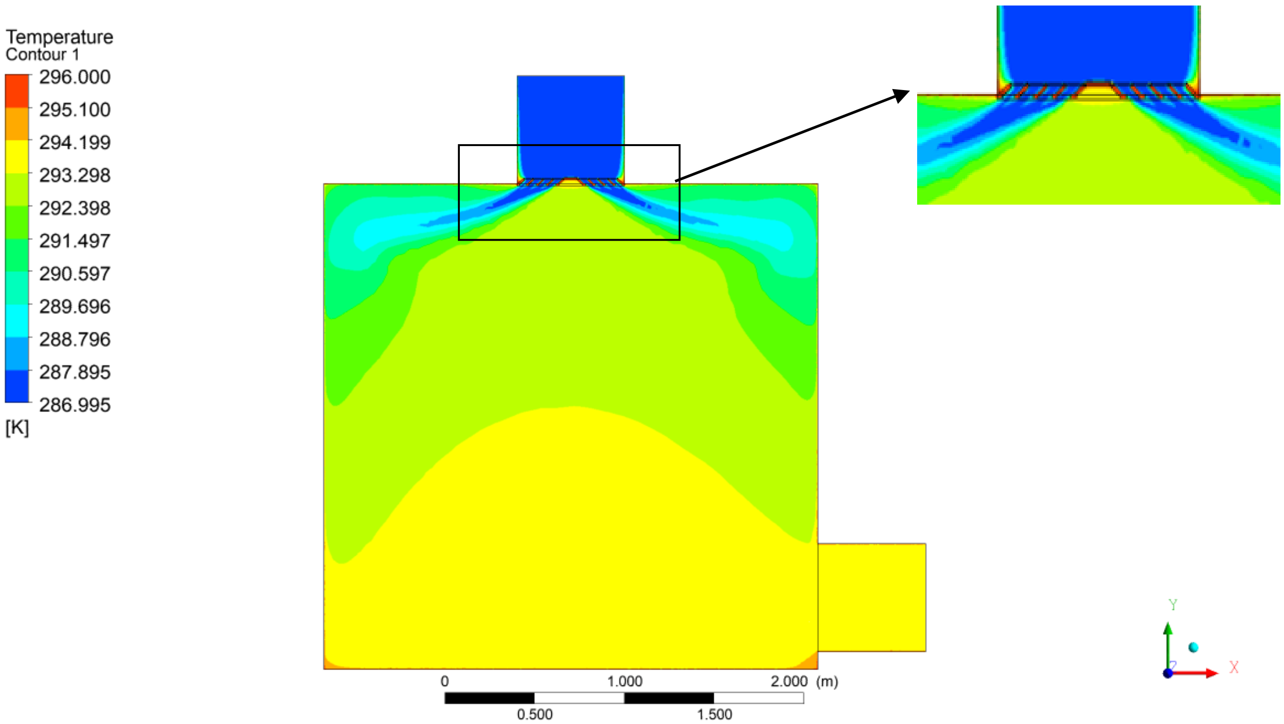


Fig. 8. Static temperature of the air flow at x-y plane (2D cut) in the center of the domain, Case II.

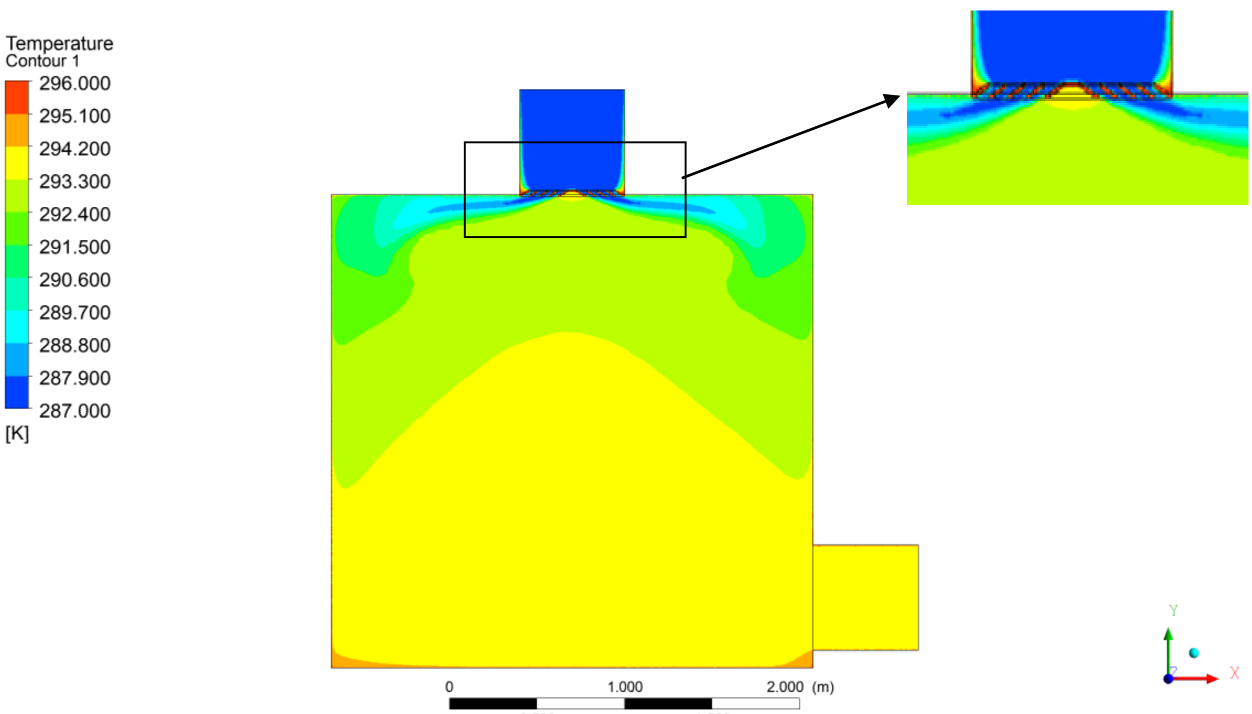
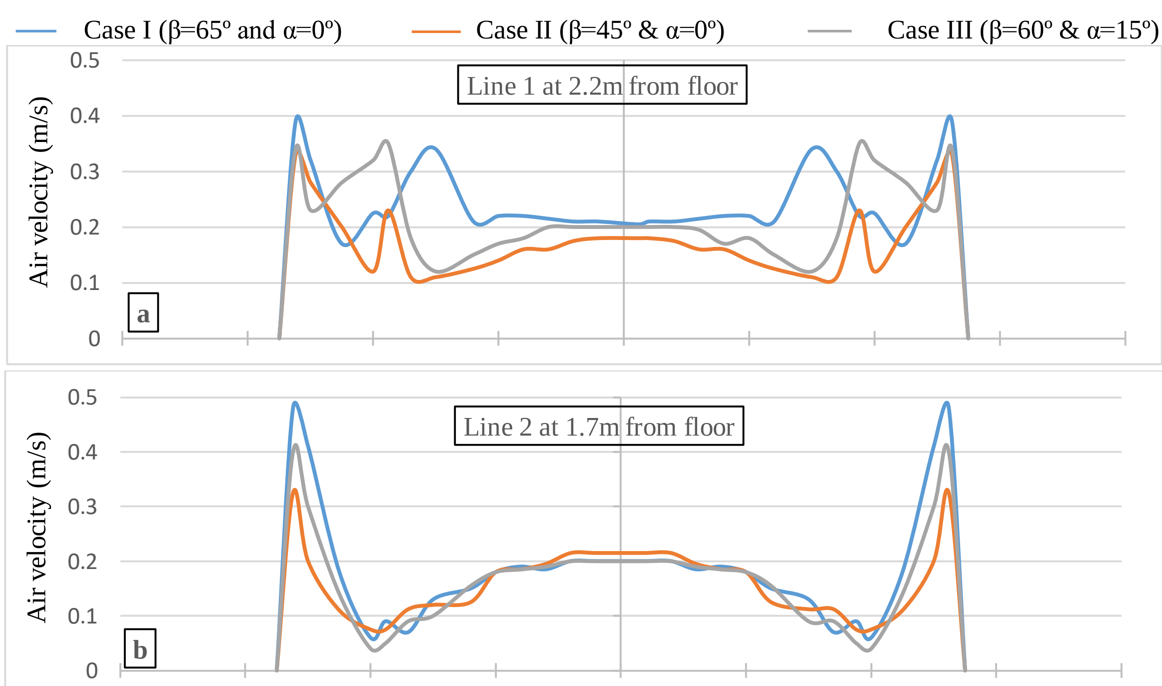


Fig. 9. Static temperature of the air flow at x-y plane (2D cut) in the center of the domain, Case III.

3.2 Air velocity and temperature distribution of new diffuser configurations

Figures 10 and 11 show a comparison between the investigated diffuser configurations (Case I, Case II, and Case III) at different heights (see Fig. 3) inside the considered room size (see Fig. 1) and the results listed in Table 1. As shown in Fig. 10, as the blade angle of case I increases, the exit velocity of the diffuser (jet) increases, so the terminal velocity is higher than that of case II and case III at different line positions. In addition, because the airflow of Case II is conical, the jet velocity is small, and there is no attachment or adhesion to the ceiling, so the terminal velocity is relatively close to Case I and Case III. Moreover, at all different line's locations the terminal velocity from Case I and Case III is approximately the same and higher than Case II.

On the other hand, an increase in the blade angle (case I) causes the airflow from the diffuser to stick to the ceiling and increase friction, so the temperature of Case I is higher than that of Case II and Case III as shown in Fig. 11. From Case II, the conical air outlet reduces the friction between the airflow and the ceiling of the room, realizing low-temperature flow close to the wall and throughout the room. Also, Case III shows relatively the same temperature results as Case I, but the temperature results of Case III near the floor show lower temperatures than those shown in Case I due to the reduced flow friction with the ceiling in Case III. As shown in Table 1, the temperature difference between different lines/heights is lower in Case III than in Case I.



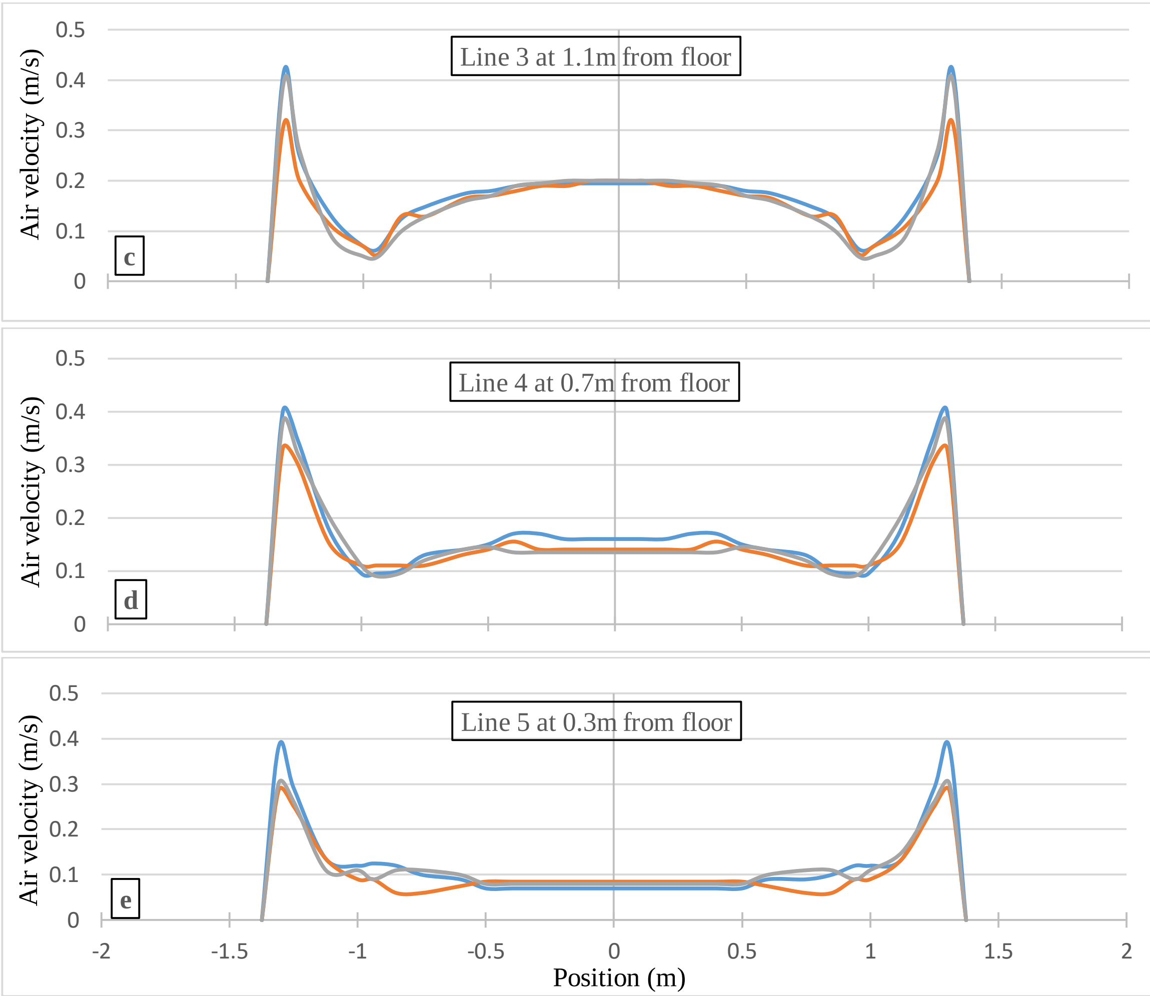
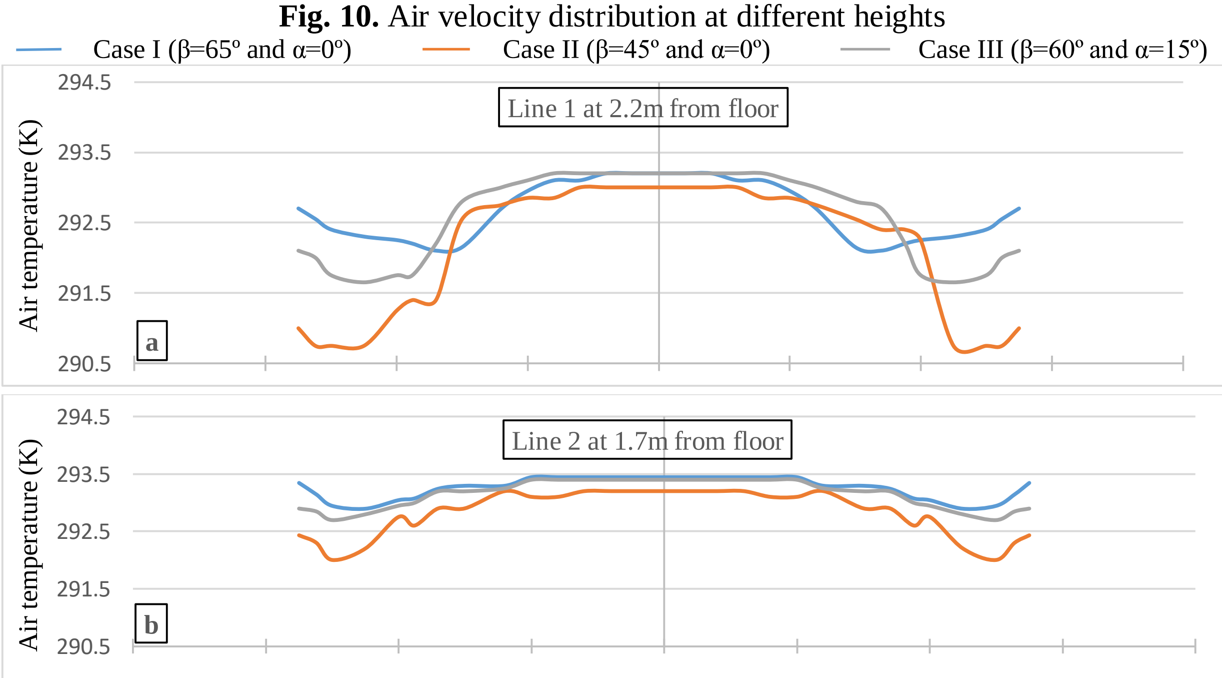


Fig. 10. Air velocity distribution at different heights



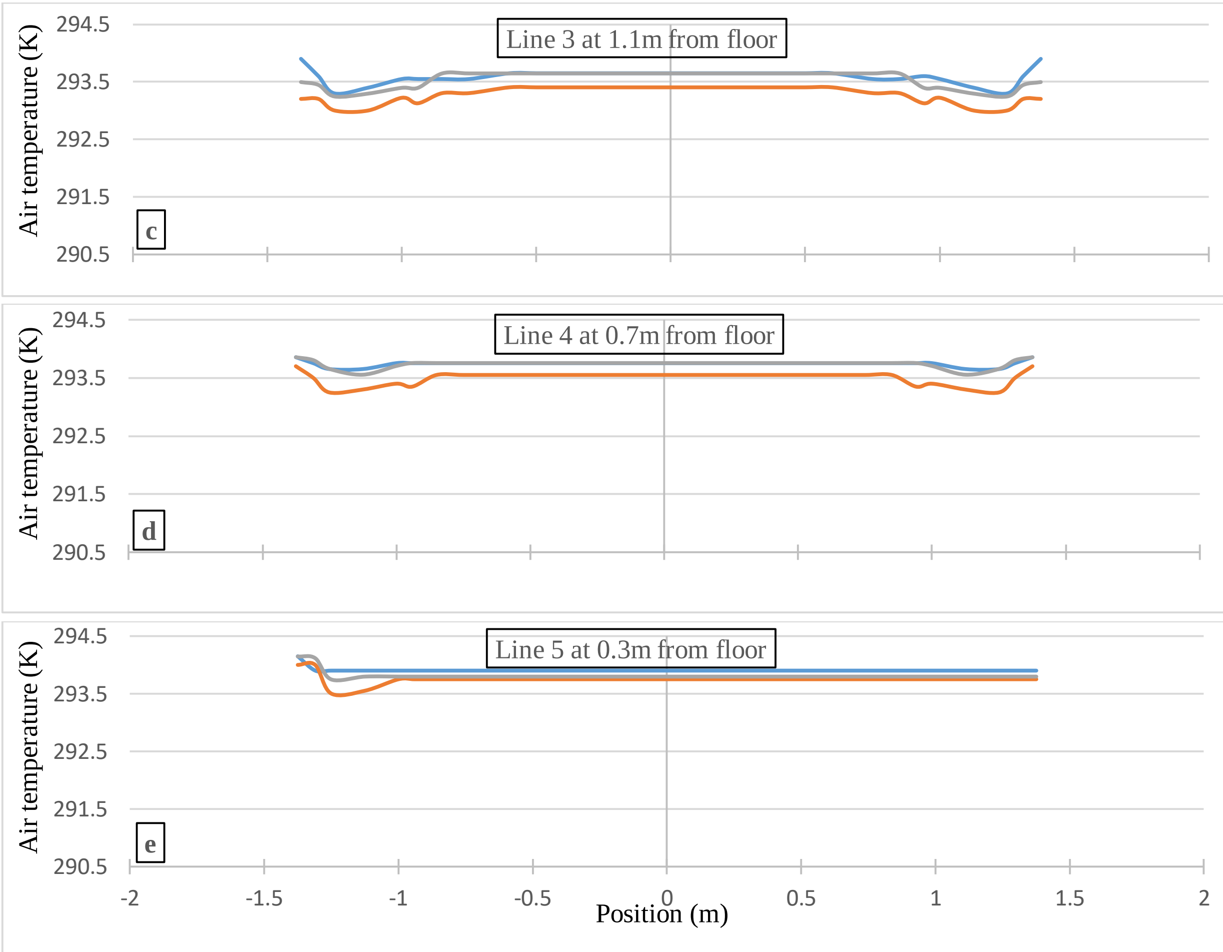


Fig. 11. Air temperature distribution at different heights

3.3 Impact of change the room size

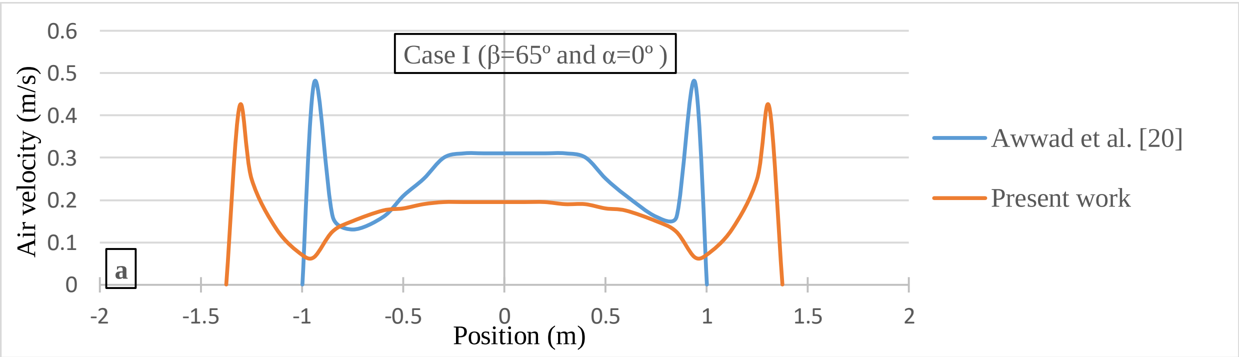
Figure 12 and Table 2 show the comparison of the current work (room dimensions 2.75 x 2.75 x 2.7 m) with the published work with room dimensions of 2 x 2 x 2 m from Awwad et al. (2017), in terms of the average amplitude velocity (terminal) for a human sitting position (0.9 m to 1.1 m from the room floor). Its range shows a higher temperature distribution than in Awwad e.al. [20]. This is due to the velocity reduction discussed above. Lower velocity means less convective heat transfer resulting in higher temperatures. The results of the three cases are relatively similar., with little difference in the small space. This effect will be noticed even in large spaces using a few new diffuser configurations, where the number of diffusers is reduced, reducing duct work (reducing costs), and achieving thermal comfort more quickly. Furthermore, Table 2 shows the comparison of the average velocity values for Cases I, II, and III with those shown in the previous study [20] for human sitting height. All values are in range and consistent with standard comfort conditions (below 0.25 m/s), except for the case I in a small room, where the value is above 0.25 m/s. Thus, in both the current work in large rooms and the published work in small rooms [20], the temperature and velocity profiles of the new diffuser conform to the minimum recommended values of standard and achieve thermal comfort conditions.

DOI : https://doi.org/10.56452/2023-04/003

4. Conclusions

In this paper, ANSYS software is used to develop a simulation model of a novel ceiling air diffuser (LFCD) with different lip and blade angles in a room of size 2.75 x 2.75 x 2.7 m. The thermal comfort criteria in the room were examined with the temperature and velocity profiles of three different configurations of the novel ceiling air diffuser (LFCD). The reported results based on the present work reveal the following conclusions.

- The average terminal velocity values for a blade angle of 65° and a lip of 0° (Case I) are in the range of 0.16 – 0.21 m/s and around 0.12 m/s near the room floor. These values are consistent with the recommendations of standard guidelines (maximum 0.25 m/s, minimum 0.1 m/s near the floor of the room).
- The novel ceiling air diffuser (LFCD) with a blade angle of 65° and a lip of 0° (Case 1) has a temperature difference across the room of 1.64°C, which is below the standard recommended maximum value (3°C).
- The diffuser in case I can be used in the spaces with larger sizes to achieve comfort conditions faster than the conventional model.
- The novel ceiling air diffuser (LFCD) with a blade angle of 45° and a lip of 0° (Case II) has the average terminal velocity across the room in the range of 0.15 – 0.17 m/s, and 0.1 m/s near the room floor which in line with standard guides.
- A temperature difference of 1.65°C was achieved using a novel ceiling air diffuser (LFCD) with a blade angle of 45° and a lip of 0° (Case II).
- The average terminal velocity of the novel diffuser (Case III) with a blade angle of 60° and a lip angle of 15° across the room is in the range of 0.15 – 0.20 m/s and is about 0.11 m/s near the room floor.
- For the novel ceiling air diffuser (Case III) with a blade angle of 60° and a lip angle of 15°, the temperature difference across the room is close to 1.24°C, which is below the maximum value of the recommended standard.
- In general, the three new diffuser configurations achieved the standard thermal comfort conditions in the investigated rooms.



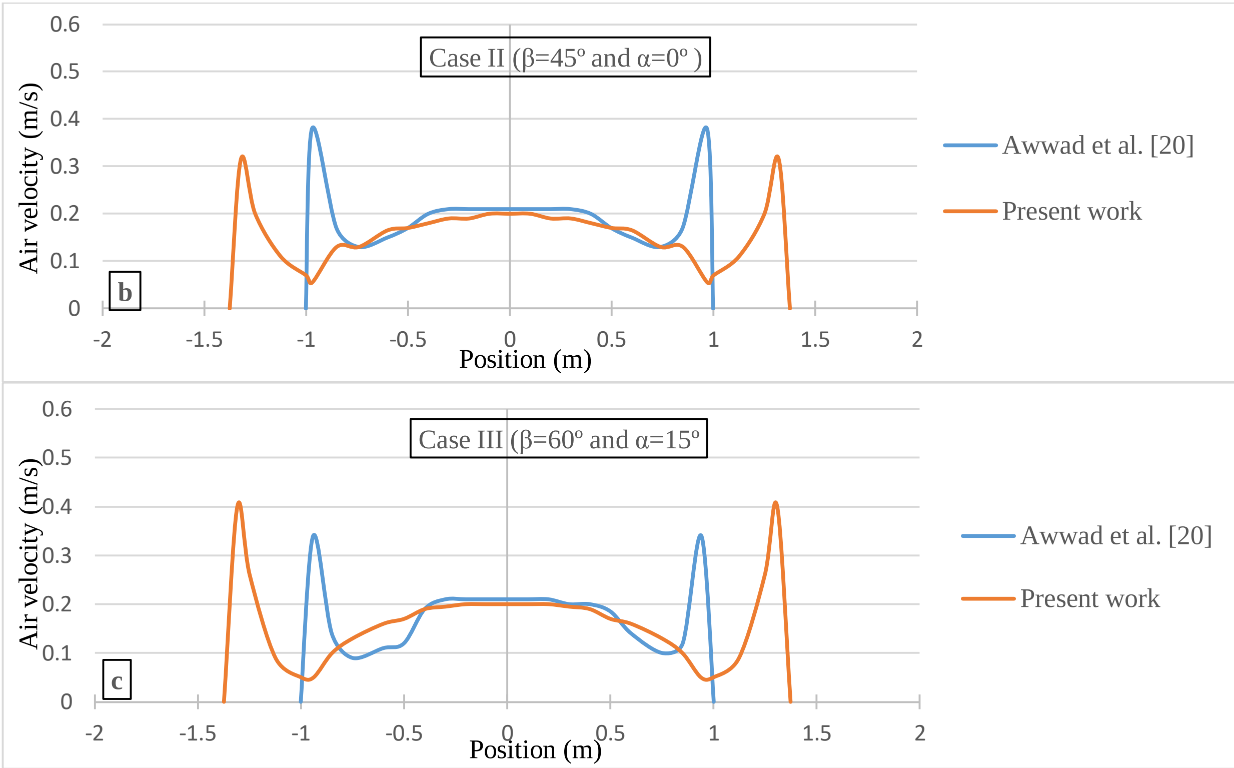


Fig. 12. Air velocity distribution at human sitting position (line 3 at 1.1m from room floor)

Table 2 Average of air velocity at human sitting position

Case no.	Line no.	Position from room floor (m)	Average of air velocity magnitude (m/s)	
			Awwad et al. [20]	Present work
Case I	Line 3	1.1 m	0.263	0.170
Case II			0.204	0.153
Case III			0.200	0.159

Reference

[1] ASHRAE handbook of fundamentals, Chapter 20, Space Air Diffusion, American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc., 2021.

[2] R. H. Mohammed, A simplified method for modeling of round and square ceiling diffusers, Energy and Buildings 64 (2013) 473–482, DOI: 10.1016/j.enbuild.2013.05.021.

[3] Y. Sun, F. Smith, Air flow characteristics of a room with square cone diffusers, Building and Environment 40 (2005) 589–600, DOI: 10.1016/j.buildenv.2004.07.018.

[4] M.A. Aziz, I.A.M. Gad, E.S.F.A. Mohammed, R.H. Mohammed, Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics, Energy and Buildings 55 (2012) 738–746, DOI: 10.1016/j.enbuild.2012.09.027.

[5] E. Djunaedy, K.W.D Cheong, Development of a simplified technique of modelling four-way ceiling air supply diffuser, Building and Environment 37 (2002) 393–403.

[6] Tavakkol S, Hosni MH, Miller PL, Straub HE. A study of isothermal throw of air jets with various room sized and outlet configurations. ASHRAE Transactions 1994;100(1):1679–86.
[7] J. Abanto, D. Barrero, M. Reggio, B. Ozell, Airflow modelling in a computer room, Building and Environment 39 (2004) 1393-1402, DOI: 10.1016/j.buildenv.2004.03.011.
[8] A. Fontainni, M.G. Olsen, B. Ganapathysubramanian, Thermal comparison between ceiling diffusers and fabric ductwork diffusers for green buildings, Energy and Building 43 (2011) 2973-2987, DOI: 10.1016/j.enbuild.2011.07.005
[9] J.D. Ponser, C.R. Buchanan, D. Dunn-Rankin, measurement and prediction of indoor air flow in a model room, Energy and Building 35 (2003) 515-526.
[10] Z.F. Tina, J.Y. Tu, G.H. Yeoh, R.K.K. Yuen, On the numerical study of contaminant particle concentration in indoor airflow, Building and Environment 41 (2006), 1504-1514, DOI: 10.1016/j.buildenv.2005.06.006
[11] Z.F. Tina, J.Y. Tu, G.H. Yeoh, R.K.K. Yuen, Numerical studies of indoor airflow and particle dispersion by large eddy simulation, Building and Environment 42 (2007), 3483-3492, DOI: 10.1016/j.buildenv.2006.10.047.
[12] W. Lu, A.T. Howarth, N. Adam, S.B. Riffat, Modeling and measurement of airflow and aerosol particle distribution in a ventilated two-zone chamber. Building and Environment 31. (1996), 417–23.
[13] J. Fan, C. A. Hviid, H. Yang, Performance analysis of a new design of office diffuse ceiling ventilation system, Energy and Buildings 59 (2013), 73-81. DOI: 10.1016/j.enbuild.2013.01.001.
[14] T. Zhang, K. Lee, Q. Chen, A simplified approach to describe complex diffusers in displacement ventilation for CFD simulations, Indoor Air 2009; 19: 255–267. DOI: 10.1111/j.1600-0668.2009.00590.x.
[15] Y. Bin, S.C. Sekhar, Three-dimensional numerical simulation of a hybrid fresh air and recirculated air diffuser for decoupled ventilation strategy, Building and Environment 42 (2007) 1975–1982.
[16] S. Liu, A.Novoselac, Air Diffusion Performance Index (ADPI) of diffusers for heating mode, Building and Environment 87 (2015), 215-223. DOI: 10.1016/j.buildenv.2015.01.021.
[17] ASHRAE standard 70-2006 (RA 2021), Method of testing and the performance of air outlets and air inlets.
[18] S. Liu, A. Novoselac, Lagrangian particle modeling in the indoor environment: a comparison of RANS and LES turbulence methods (RP-1512). HVAC&R Res 2014;20(4):480e95.
[19] M. Vasic, V. D. Stevanovic, B.Zivkovic, Uniformity of air flow from the ceiling diffuser by an advanced design of the equalizing element in the plenum box with side entry, Science and Technology for the Built Environment (2020), ISSN: 2374-4731.
[20] A. Awwad, M. H. Mohamed, M. Fatouh. Optimal design of a louver face ceiling diffuser using CFD to improve occupant's thermal comfort, Journal of Building Engineering 11 (2017) 134–157. DOI: 10.1016/j.jobe.2017.04.009.
[21] International Building Code (IBC) 2021, Section 1208 – Interior Space Dimensions.
[22] Ansys Inc., Ansys Fluent release 15.0, Canonsburg, PA, USA, November 2013.
[23] ASHRAE standard 55-2020, Thermal environmental conditions for human occupancy.
[24] SMACNA HVAC system – duct design, chapter 3 - room air distributions.