Düzce University Journal of Science and Technology
Düzce Üniversitesi Bilim ve Teknoloji Dergisi

14 (3), 2026, 786–797.
https://doi.org/10.29130/dubited.1846844

e-ISSN: 2148-2446
https://dergipark.org.tr/en/pub/dubited

Research Article

Investigation of Indoor Air Temperature and Velocity Distribution in an Office at Different Inlet Conditions Using CFD

Hande Ufat[1*]

[1] Bursa Uludag University, Vocational School of Technical Sciences Air Conditioning and Refrigeration Program, Bursa, Türkiye.
ror.org/03tg3eb07

*Corresponding author:
handet@uludag.edu.tr

Received: 22.12.2025
Revision: 23.04.2026
Accepted: 11.05.2026
Online: 24.07.2026

Abstract: This paper presents, the distribution of indoor air temperature and velocity in an office environment under different inlet conditions using Computational Fluid Dynamics (CFD). The office, with dimensions of 8 × 5 × 3 m, was cooled by a wall-mounted split air conditioner. The analyses were performed using the k-ε turbulence model, with the split air conditioner's inlet temperature set to 18, 20, and 22°C and its velocity set to 0.5, 1, and 2 m/s. The analysis included the heat emitted by five people, five computers, and lamps in the office. The results indicate that increasing inlet velocity leads to a more uniform temperature distribution and lower average indoor temperatures. At an inlet velocity of 0.5 m/s, insufficient air mixing resulted in localized temperature variations, while at 2 m/s, local air velocities in certain regions exceeded 0.25 m/s. According to ASHRAE Standard 55, this value represents the upper limit for acceptable air velocity in occupied zones and may lead to draft risk. At an inlet velocity of 1 m/s, a more balanced distribution of temperature and air velocity was achieved, with indoor temperatures generally within the recommended range of 22–26°C and air velocity remaining mostly below the threshold value. The findings demonstrate that inlet air conditions significantly influence indoor air distribution characteristics. Among the evaluated cases, an inlet velocity of 1 m/s provided the most favorable balance between temperature distribution and air velocity within the investigated conditions.

Keywords: *Temperature and air velocity distribution, Split air conditioning, Cooling, CFD*

I. INTRODUCTION

Today, many people work in closed offices and spend most of their time there. Indoor environmental conditions is crucial for the health and work performance of employees. Among these conditions, air temperature and air velocity are key parameters influencing indoor air distribution and overall environmental quality. In this context, numerical methods such as Computational Fluid Dynamics (CFD) have become widely used tools for analyzing airflow behavior and temperature distribution in indoor spaces. The fundamental principles of indoor environmental conditions and human response were established in early studies, such as Fanger (1970), which highlighted the relationship between environmental parameters and human perception. In addition, Nielsen (1998) emphasized the importance of selecting appropriate turbulence models for accurately predicting indoor airflow, while Chen (2009) provided a comprehensive overview of ventilation performance prediction methods and demonstrated the effectiveness of CFD in indoor environment analysis. These foundational studies indicate that airflow distribution in enclosed spaces is highly sensitive to inlet conditions, boundary definitions, and modelling approaches.

Subsequent research has focused on improving numerical modeling techniques and understanding the impact of air supply parameters on indoor environments. Awbi (2003) reported that supply air characteristics, particularly inlet velocity and temperature, significantly affect airflow patterns and thermal stratification in buildings. Various studies have proven that improving thermal comfort parameters increases work performance (Ganesh et al., 2024; Othayq, 2025; Tang et al., 2021). Furthermore, CFD analyses can significantly reduce invest ment costs by allowing the effects of improvements to be made to the existing system or the efficiency of a system to be designed to be seen and evaluated before implementation. Recent CFD-based research has also investigated indoor airflow distribution and ventilation performance under different operating conditions. For example, Zhang and Ryu (2021) analyzed indoor air and humidity distribution under various ventilation strategies, demonstrating that airflow patterns significantly affect both thermal conditions and indoor environmental quality. More recently, Omerzo et al. (2025) conducted a study combining experimental and CFD to investigate the effect of air distribution systems and airflow velocities on the transmission of airborne pollutants, highlighting the critical role of airflow structure in determining indoor air quality and exposure levels. These studies reveal that ventilation parameters and airflow characteristics are key factors influencing both the thermal environment and pollutant transport in indoor spaces. Lu et al. (2021). This study examines the use of

Cite as: Ufat, H. (2026). Investigation of indoor air temperature and velocity distribution in an office at different inlet conditions using CFD. *Düzce University Journal of Science and Technology, 14*(3), 786–797. https://www.doi.org/10.29130/dubited.1846844

This is an open access paper distributed under the terms and conditions of the Creative Commons Attribution-NonCommercial 4.0 International License.

computational fluid dynamics (CFD) simulations to ensure thermal comfort and increase energy savings. The effects on airflow, temperature distribution, and human comfort in different indoor scenarios were analysed. Through CFD-based modelling, strategies were developed to ensure the efficient operation of HVAC systems, resulting in significant improvements in energy consumption. The study demonstrates that CFD methods are an effective tool both in the design phase and in the optimisation of existing systems. Horikiri et al. (2015) highlighted the influence of occupant presence, ventilation strategies, and room configuration on airflow behavior and thermal conditions.

Höfert and Danielak (2021) use CFD (Computational Fluid Dynamics) analyses to compare three different ventilation approaches in terms of pathogen spread and thermal comfort in open-plan office environments. The systems compared are mixed ventilation, displacement ventilation, and ceiling diffuser ventilation. Simulation results revealed that some systems were more effective than others in reducing pathogen concentration and providing more homogeneous thermal comfort. The study emphasises the importance of appropriate ventilation strategies in improving indoor air quality and reducing the risk of infection. Huang et al. (2025) conducted simultaneous heat and mass transfer analyses to evaluate indoor air quality and thermal comfort in naturally ventilated offices. Using CFD-based modelling, parameters such as temperature distribution, humidity, and air movement were examined in detail. The results show that when properly designed, natural ventilation can increase indoor air quality and support thermal comfort conditions by providing an adequate air exchange rate. This comprehensive approach provides important findings for balancing energy efficiency with healthy indoor environmental conditions.

Split air conditioners are also the most suitable systems in places without a central air conditioning system. Provided that fresh air is supplied through open windows, they offer a suitable solution for ensuring thermal comfort. In addition to placing the split air conditioner in the most suitable position, ensuring the optimum inlet temperature and speed during use is important both for the comfort of those working indoors and for energy efficiency. Considering the energy consumption of air conditioning systems, there is a great deal of research in this field aimed at improving work performance by increasing the thermal comfort of working environments, as well as saving energy.

Experimental and numerical studies have also confirmed that CFD models can provide reliable predictions when validated against experimental data (Catalina et al., 2009). Furthermore, studies focusing on natural and mechanical ventilation systems have shown that airflow characteristics directly influence indoor environmental quality and energy performance (Fan et al., 2022; Mora-Pérez et al., 2017). These findings indicate that air velocity and temperature are critical parameters in evaluating indoor airflow performance. Tang et al. (2021) investigated how portable cooling systems affect both thermal comfort and work performance in areas without air conditioning or where people are moving around, i.e., areas defined as 'hot environments.' The physiological and behavioural outcomes of different applications were evaluated at an air temperature of 32°C. Four conditions were examined: directing cold air to the breathing zone, directing cold air to the chest and back, combining these two systems, and applying no cooling at all. Cooling the chest and back resulted in increases in work performance of approximately 17% and 19%, respectively, while combining the two systems resulted in an increase of approximately 33%. A decrease in skin temperature and heart rate was observed, along with a reduction in stress. Baharudin et al. (2024) conducted a CFD analysis study to make the temperature distribution in an empty office more homogeneous and to save energy using split air conditioning and double panel radiators. By placing split air conditioners at different points and performing analyses, a more homogeneous air distribution and energy savings were achieved. However, the fact that people were not modelled in the environment is a shortcoming of this study because even if there are few people in relatively small environments such as offices, the heat they emit will affect the temperature distribution in the environment. Mazlan et al. (2024) evaluated the performance of an existing air-conditioning system in a medium-sized office building using the Computational Fluid Dynamics (CFD) method, focusing on temperature distribution imbalances and energy inefficiencies. In the model developed with Ansys software, the k-ε turbulence model was employed to simulate different air-conditioner placements. In this way, the influence of air-conditioner location on the indoor temperature distribution was emphasized. Hatif et al. (2020) investigated the effects of different air supply and exhaust positions on air distribution and thermal comfort in a small office environment using a CFD program. In their study, four different supply–exhaust layout scenarios were evaluated, and for each case, air temperature distribution, velocity profiles, Air Distribution Performance Index (ADPI), and ventilation effectiveness were analyzed. The results showed that placing exhaust outlets near heat sources (such as occupants, computers, and lighting) improved thermal comfort.

Muhamad et al. (2022) analyzed airflow and temperature distribution in an office environment using CFD by comparing two different layout scenarios. In this study, the effects of air supply and exhaust locations on indoor comfort conditions were examined in detail, particularly under a displacement ventilation strategy. The results indicated that the first scenario, where the exhaust outlet was positioned farther from the air supply, provided better air distribution performance.

The literature contains numerous studies examining airflow, ventilation performance, and thermal comfort in indoor environments using CFD methods. These studies generally focus on temperature distribution, airflow characteristics, or comfort indices under specific operating conditions. In addition, several studies have investigated the performance of air-conditioning systems by considering parameters such as device placement, airflow direction, and ventilation strategy.

More specifically, studies on split-type air conditioners remain relatively limited compared to those on other

ventilation systems. For example, Zhang et al. (2022) investigated the optimal placement and air distribution characteristics of split floor standing type air conditioners, mainly emphasizing airflow structure and positioning effects. Similarly, Pulat and Yuce (2021) conducted a detailed CFD-based parametric study to evaluate the influence of inlet air temperature and velocity on thermal comfort, ventilation performance, and indoor air quality using a breathing thermal manikin model in a realistic office geometry. Their results indicated that different combinations of inlet parameters can yield similar thermal comfort levels despite variations in temperature and velocity, highlighting the complex interaction between airflow and heat transfer mechanisms. Although these studies provide valuable insights, they also have certain limitations. Previous research has generally focused on either comfort indices such as PMV/PPD or specific parameters such as airflow direction and device positioning, while the combined spatial evaluation of temperature distribution and airflow velocity structure within the occupied zone has received comparatively less attention. Furthermore, many studies either emphasize indoor air quality (e.g., CO_2 distribution) or thermal comfort metrics, without simultaneously examining airflow-induced discomfort, such as draft risk, in relation to velocity limits.

Therefore, a clear research gap exists in terms of conducting an integrated assessment that simultaneously considers inlet air parameters, thermal distribution, airflow structure, and draft risk under realistic indoor conditions. In this context, the present study aims to fill this gap by performing a comprehensive CFD analysis of an office environment equipped with a wall-mounted split air conditioner. Unlike previous studies, the model incorporates internal heat sources such as occupants, computers, and lighting, enabling a more realistic representation of indoor conditions. Moreover, the study evaluates the coupled effects of inlet air temperature and velocity on both temperature distribution and airflow characteristics, while also assessing draft risk based on air velocity criteria (American Society of Heating, Refrigerating and Air-Conditioning Engineers, 2020). Consequently, this study provides a more integrated and practically applicable framework for optimizing air-conditioning performance in office environments.

II. MATERIAL AND METHOD

A. *Physical Model and Case Description*

The office was modeled in the ANSYS Workbench program. The dimensions of the office are 5 × 8 × 3 m. The window and the wall containing the window are connected to the outdoor environment. The wall where the split air conditioner is mounted is adjacent to another office with similar conditions, while the other two walls are adjacent to the production area and are entirely glass surfaces. The wall shared with the neighboring office, as well as the floor and ceiling, were assumed to be adiabatic. The modeled office is shown in Figure 1. The occupants in the office were represented as cylinders, and there were five in total. In addition, each occupant was assigned a desk and a laptop. Surface temperature values were specified for both the laptops and the occupants during the analysis.

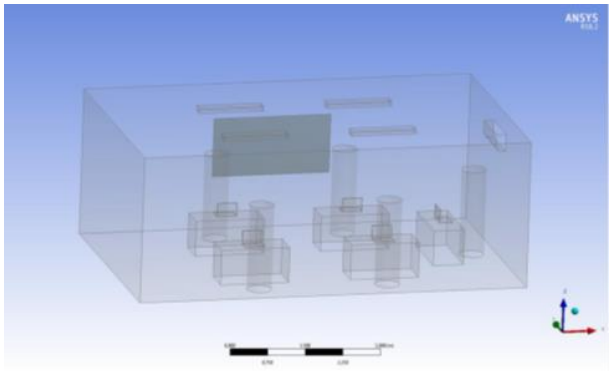


Figure 1. Office model.

B. *Numerical Method*

The standard k–ε model and the standard wall function were used as the computational tools. The simulation was conducted in steady-state condition. The transport equations of the standard k–ε turbulence model (Ufat et al., 2017) can be written as follows:

The continuity:

$$\nabla(\rho\vec{v}) = S_m \tag{1}$$

Momentum:

$$\nabla.(\rho\vec{v}\,\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} \tag{2}$$

Here, $\bar{\bar{\tau}}$ represents the viscous stress tensor and can be expressed as: $\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I\right]$

Energy equations:

$$\nabla . \left(\vec{v} \left(\rho \vec{v} + p\right)\right) = -\nabla . \left(\sum_j h_j J_j\right) + S_h \quad (3)$$

Turbulent kinetic energy:

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (4)$$

Dissipation rate of turbulent kinetic energy:

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + G_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (5)$$

Turbulent viscosity was calculated using the following relation:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

C_μ, $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k and σ_ε are constants and their values are C_μ=0.09, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, σ_k=1.0 and σ_ε=1.3. The term G_k, representing the production of kinetic energy, can be defined as $G_k = -\rho\overline{u_i u_j}\frac{\partial u_j}{\partial x_i}$. To evaluate this parameter in a manner consistent with the Boussinesq hypothesis, we use $G_k = \mu_t S^2$, where the modulus of the mean rate-of-strain tensor is defined as $S \equiv \sqrt{2S_{ij}S_{ij}}$.

The generation of turbulence due to buoyancy is given by $G_b = \beta g_i \frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i}$, where Pr_t is the turbulent Prandtl number for energy and g_i is the component of the gravitational vector in the *i*th direction. For the standard k-ε model the default value of Pr_t is 0.85. The coefficient of thermal expansion, β, is defined as $\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$. The dilatation dissipation term Y_M is modelled as $Y_M = 2\rho\varepsilon M_t^2$, where M_t is the turbulent Mach number; the turbulent Mach number is defined as $M_t = \sqrt{\frac{k}{a^2}}$ where α ($\equiv \sqrt{\gamma RT}$) is the velocity of sound.

C. Validation

The numerical model employed in this study is based on a previously validated CFD approach reported by Ufat et al. (2017). In that study, experimental measurements of air temperature and air velocity were conducted in an indoor environment, and the results were compared with CFD analysis using similar modelling assumptions. When one of the air velocity-altitude graphs in the referenced study is evaluated, the data in Table 1 is observed.

Table 1. Validation data of Ufat et al. (2017).

Heigh (m)	Velocity m/s (experimental)	Velocity m/s (numerical analysis)
0.5	0.22	0.2
1	0.24	0.22
1.5	0.21	0.24
2	0.21	0.22

The comparison between experimental and numerical results in the referenced study showed good agreement, indicating that the selected turbulence model, boundary condition definitions, and numerical approach are suitable for predicting indoor air distribution.

In the present study, the same modelling framework, including the standard k–ε turbulence model, similar boundary condition definitions, and steady-state assumptions, was employed. Therefore, the numerical model used in this work relies on a previously validated methodology.

In addition, the airflow and temperature distribution patterns obtained in this study were found to be consistent with the trends reported in the literature, particularly in terms of airflow structure and temperature stratification within the occupied zone.

D. Mesh Independence

To ensure the accuracy and reliability of the numerical solution, a grid independence study was performed to evaluate the sensitivity of the results to the mesh structure. Analyses were carried out using different mesh densities, and the

results were compared. When further mesh refinement did not cause any significant changes in the results, the solution was considered to have reached sufficient precision. This approach is widely adopted in CFD studies. Simulations were conducted with mesh counts of 1,035,479; 1,552,913; and 1,931,854 elements, and the velocity–height profile along a specific line in the domain was plotted (Figure 2). The difference between the results obtained using 1,552,913 and 1,931,854 elements was found to be around 3%, indicating a behavior independent of the mesh structure. Further refinements were deemed unnecessary, and analyses continued with 1,552,913 elements to shorten analysis time. The mesh model is shown in Figure 3.

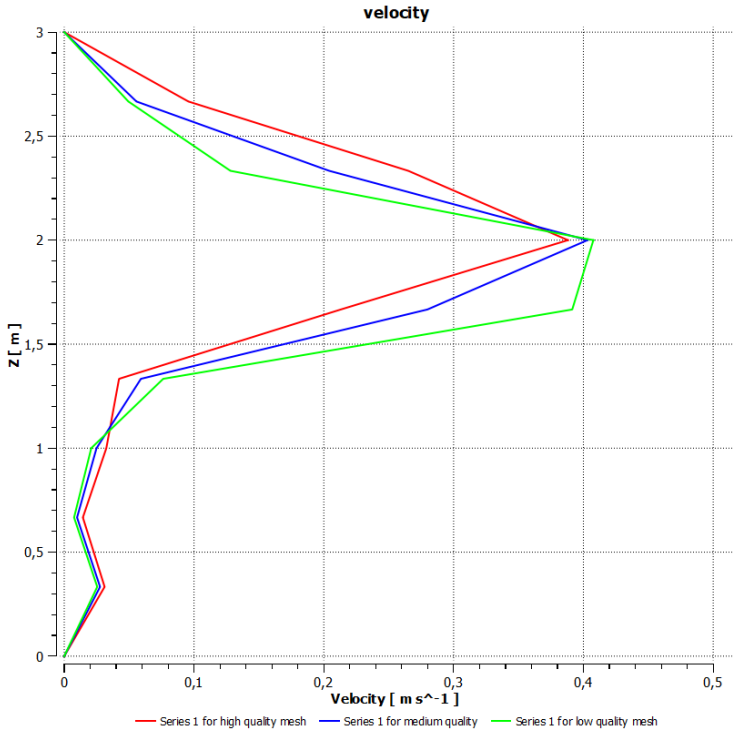


Figure 2. Height-Velocity Graph.

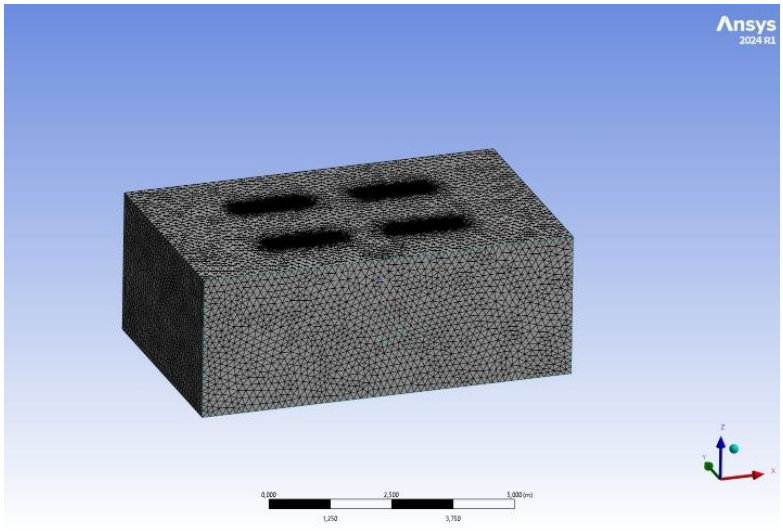


Figure 3a. Office Mesh Model.

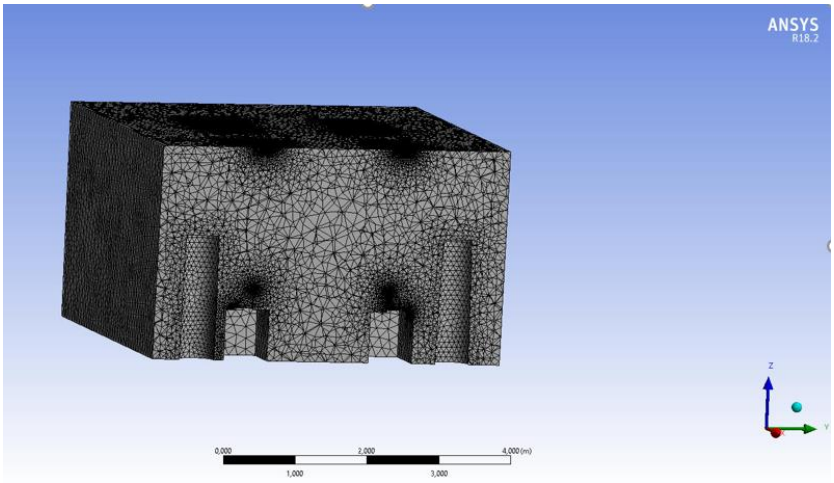


Figure 3b. Office Mesh Model Section.

E. Boundary Conditions

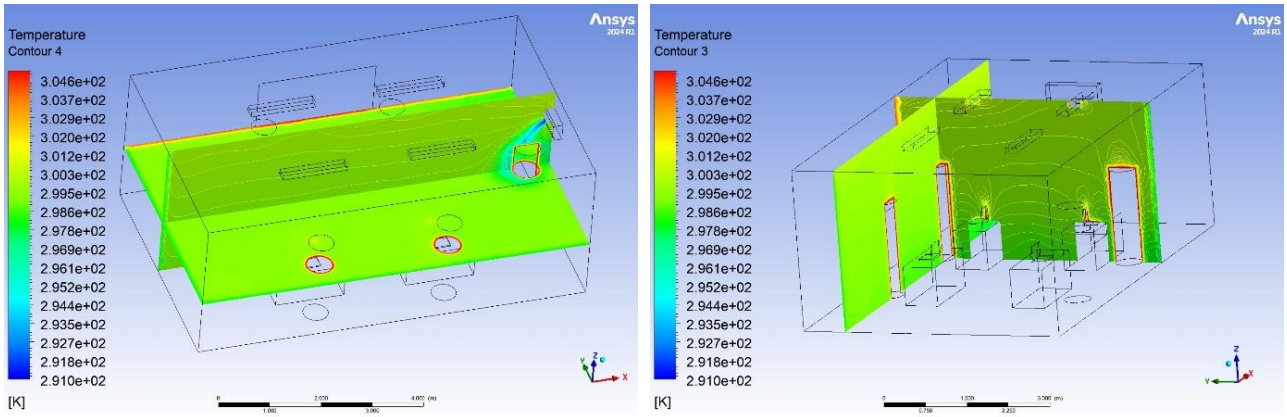
Analyses were conducted for different air inlet temperatures and velocities to determine the combination that provided the most suitable comfort conditions in the office. The simulations were performed at supply air temperatures of 18°C, 20°C, and 22°C, and inlet velocities of 0.5 m/s, 1 m/s, and 2 m/s. The standard k–ε turbulence model was used with scalable wall functions, and buoyancy effects were also included to account for density variations due to temperature differences.

The boundary conditions were defined as follows. The ceiling and floor surfaces were assumed to be adiabatic. The temperature of the production area adjacent to the interior wall was set at 24 °C. The surface temperatures of the computer, lamp, and human model were assigned as 40 °C, 33 °C, and 36 °C, respectively. The outdoor air temperature was defined as 35 °C.

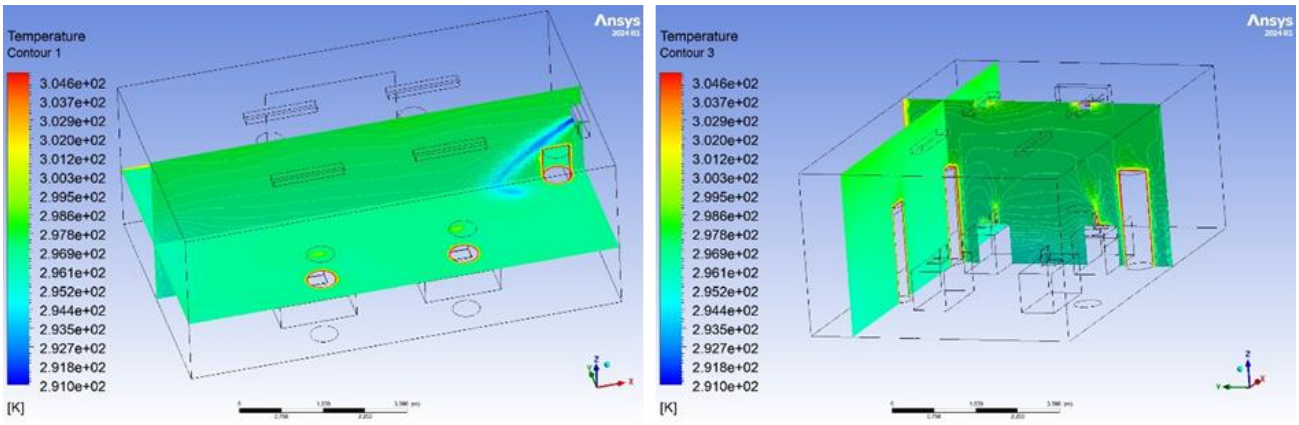
The air outlet was defined as outflow. At the inlet, the turbulent intensity was set to 5%, and the turbulent viscosity ratio to 10. The SIMPLE algorithm was used for pressure–velocity coupling. Pressure and energy equations were solved using the second-order upwind scheme, while momentum, turbulent kinetic energy, and turbulent dissipation rate were solved using the first-order upwind scheme. The convergence criteria were set to 10^{-4} for continuity, velocity components, and ε, 10^{-5} for k, and 10^{-6} for energy.

III. RESULTS AND DISCUSSIONS

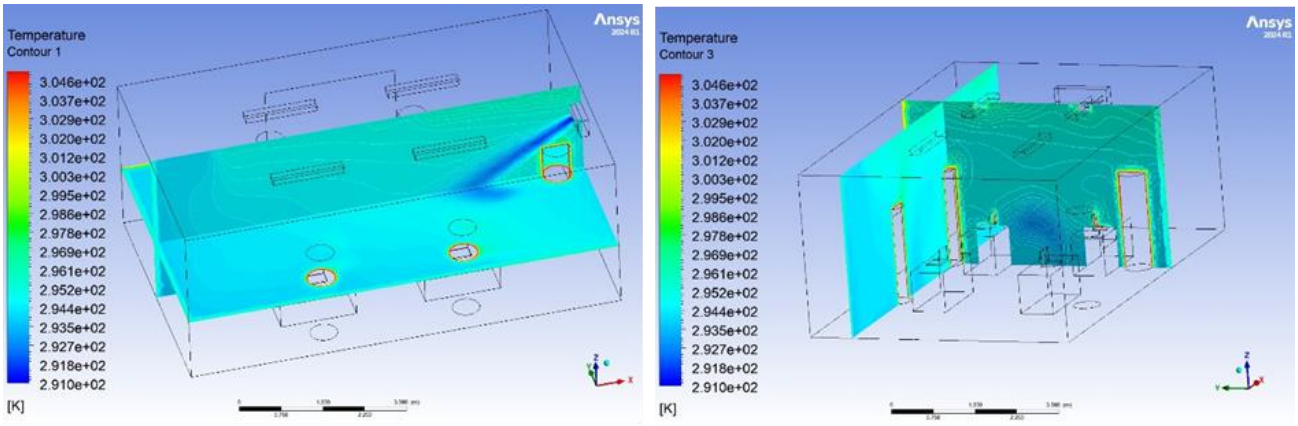
Analyses were performed for nine cases corresponding to all combinations of the specified temperatures and velocities. Temperature distributions on various cross-sections within the office were obtained for each case.



Figures 4 a – b. Temperature distribution at different sections under inlet conditions of 18°C and 0.5 m/s.

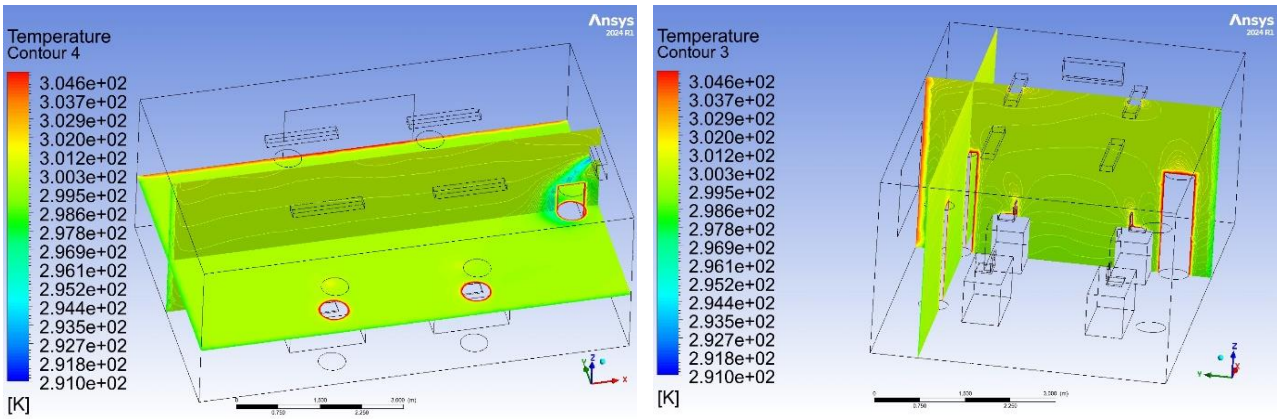


Figures 5 a – b. Temperature distribution at different sections under inlet conditions of 18 °C and 1 m/s.

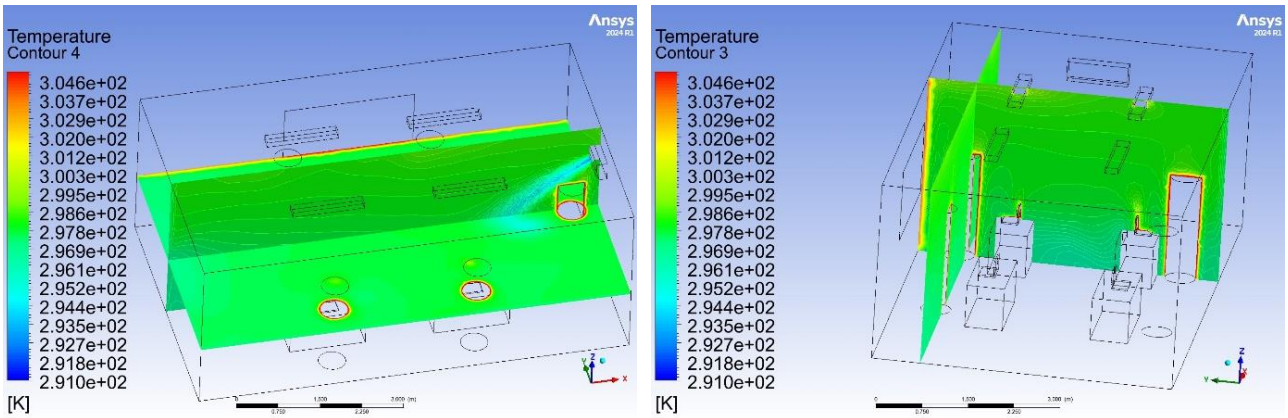


Figures 6a –b. Temperature distribution at different sections under inlet conditions of 18 °C and 2 m/s.

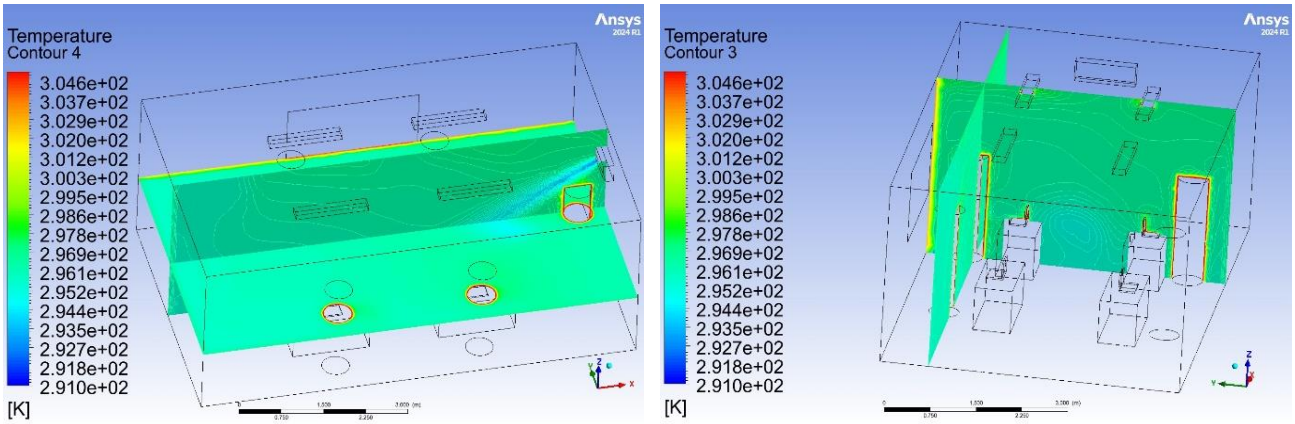
Figure 4, figure 5 and figure 6 present, when the split air conditioner supplied air at 18°C and 0.5 m/s, the average room temperature was approximately 25–26°C. Because the incoming air was relatively cool and dense, and its velocity was low, it tended to descend toward the floor under its own weight, forming a localized cool region beneath the air conditioner. At 1 m/s, the average room temperature decreased to around 23–24°C, and at 2 m/s, it dropped further to 21–22°C. Higher air velocity resulted in a lower room temperature, although the region beneath the air conditioner remained closer to the room average at 1 and 2 m/s. These temperature values are considered acceptable according to ASHRAE standards for the summer months.



Figures 7 a- b. Temperature distribution at different sections under inlet conditions of 20°C and 0.5 m/s.

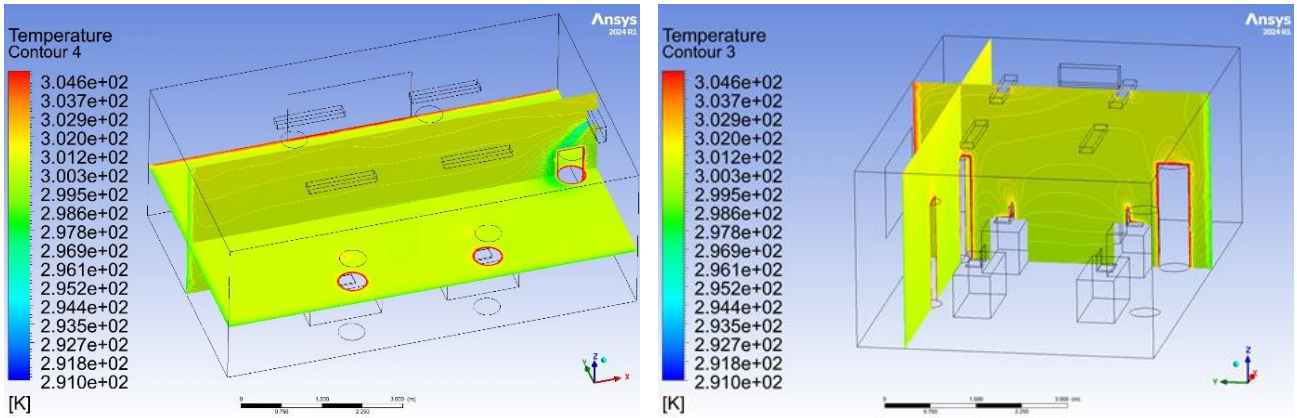


Figures 8 a – b. Temperature distribution at different sections under inlet conditions of 20 °C and 1 m/s.

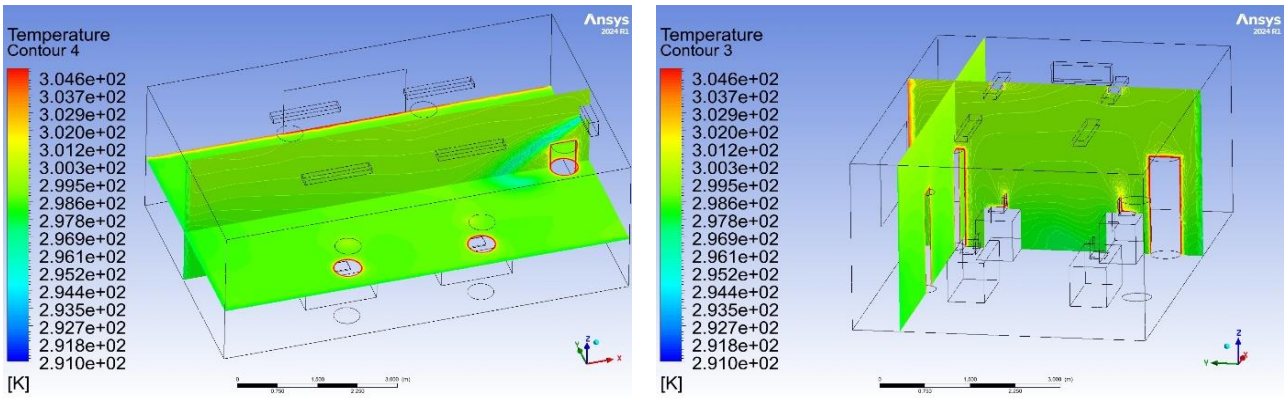


Figures 9 a- b. Temperature distribution at different sections under inlet conditions of 20 °C and 2 m/s.

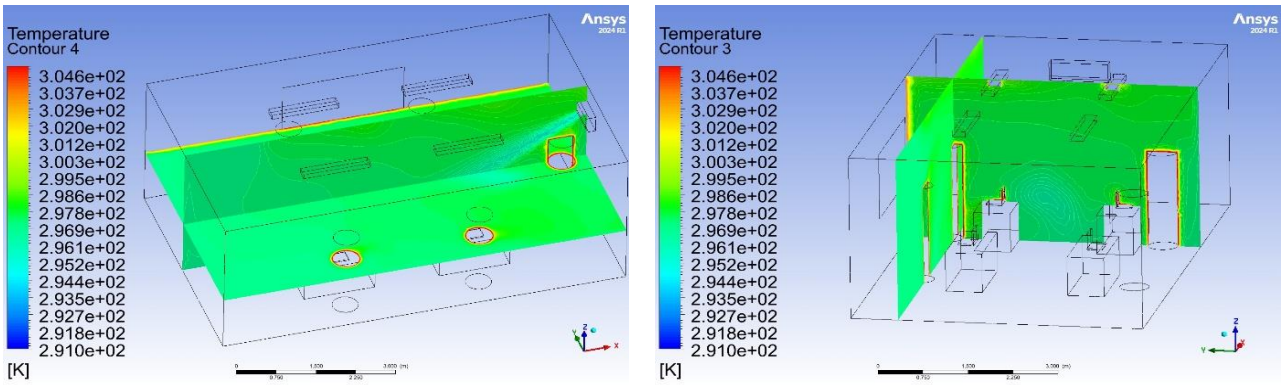
Figure 7, figure 8 and figure 9 present, when the inlet air temperature was 20°C and velocity 0.5 m/s, the average room temperature was around 26–27°C. Similar to the 18°C case, low inlet velocity caused the air to sink immediately after discharge, creating a slightly cooler zone under the air conditioner. Increasing the velocity to 1 m/s reduced the average room temperature to 23–24°C, and to 22–23°C at 2 m/s. Since temperatures around 27°C are not suitable according to ASHRAE standards, 20°C and a velocity of 0.5 m/s inlet air within the specified limit conditions is not very suitable.



Figures 10 a –b. Temperature distribution at different sections under inlet conditions of 22°C and 0.5 m/s.



Figures 11 a –b. Temperature distribution at different sections under inlet conditions of 22 °C and 1 m/s.



Figures 12 a-b. Temperature distribution at different sections under inlet conditions of 22 °C and 2 m/s.

In Figures 10, figure 11 and figure 12, at an inlet air temperature of 22°C, the same trend was observed: as the inlet velocity increased, the room temperature decreased. At 0.5 m/s, 1 m/s, and 2 m/s, the average room temperatures were 27–28°C, 25–26°C, and 23–24°C, respectively. Since temperatures around 27-28°C are higher than ASHRAE's comfort parameters under the specified boundary conditions, 22°C and 0.5 m/s velocity are unsuitable. At inlet velocities of 1 m/s and 2 m/s, the office temperature remains within standard ranges.

In all cases, the temperature near the exterior wall and window was slightly higher than the room average, as expected. However, this effect was limited to a small area. Since only five occupants and five computers were present, their total heat generation was not sufficient to significantly affect the overall temperature distribution.

In addition to temperature distribution, indoor air velocity was also evaluated, as it strongly influences indoor air temperature and velocity distribution. Even when the air temperature is within the comfort range, high local velocities can cause discomfort. The velocity streamlines on specific surfaces of the room, obtained from the analyses conducted under the aforementioned boundary conditions, are presented below.

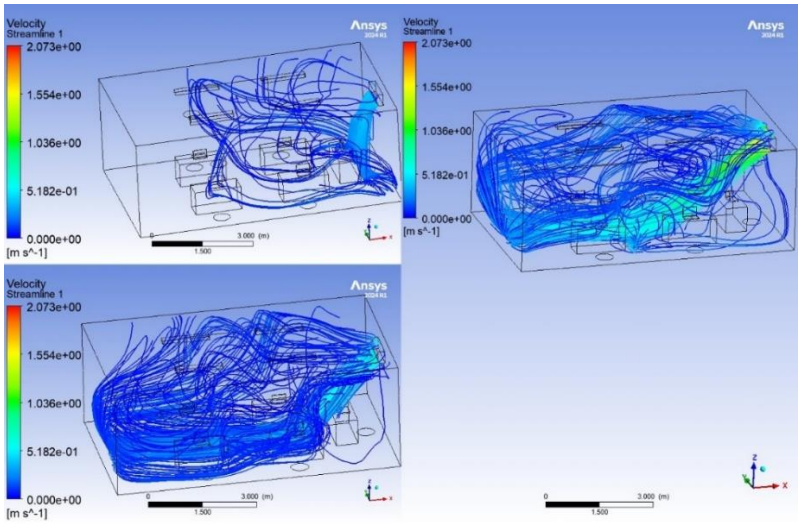


Figure 13. 18°C; Streamline for 0.5 m/s (top left), 1 m/s (down left), 2 m/s (right) inlet velocity.

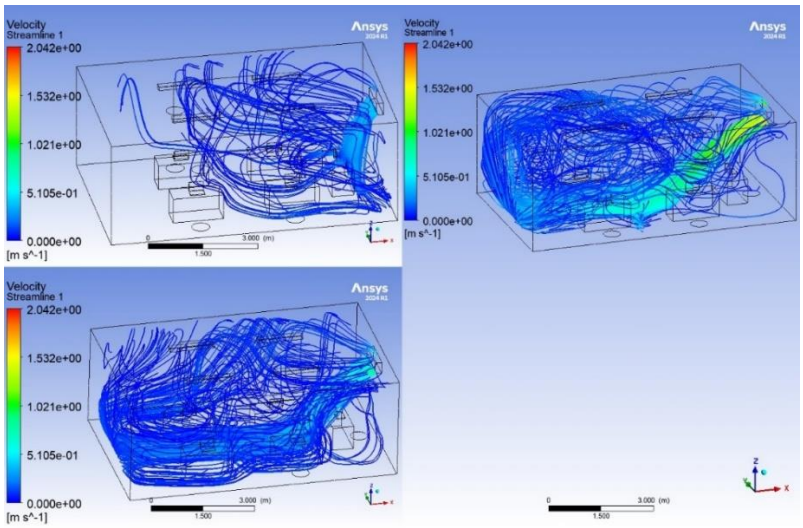


Figure 14. 20°C; Streamline for 0.5 m/s (top left), 1 m/s (down left), 2 m/s (right) inlet velocity.

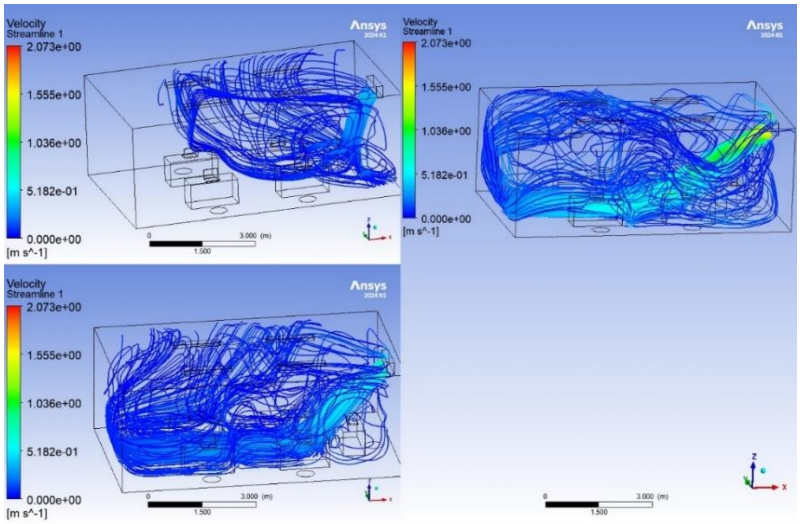


Figure 15. 22°C; Streamline for 0.5 m/s (top left), 1 m/s (down left), 2 m/s (right) inlet velocity.

Figures 13, figure 14 and figure 15 show the streamline analyses provide critical insight into the airflow characteristics and their implications for indoor air temperature and velocity distribution within the office. At an inlet velocity of 0.5 m/s, the supply air leaving the split unit exhibits relatively low momentum, causing the jet to descend rapidly and accumulate near the floor. This behavior results in insufficient air mixing and the formation of stagnant zones, particularly in the regions far from the air conditioner. Consequently, localized discomfort may occur due to limited air movement and temperature non-uniformity in these areas.

When the inlet velocity increases to 1 m/s, the airflow becomes more uniformly distributed throughout the room. Enhanced air mixing reduces the occurrence of stagnant regions and ensures that the air velocity in the occupied zone remains predominantly below 0.25 m/s, which is consistent with the acceptable range defined by ASHRAE Standard 55 American Society of Heating, Refrigerating and Air-Conditioning Engineers (2020). This velocity condition promotes both indoor air temperature and velocity distribution and effective cooling without causing draft sensations.

At 2 m/s, although the higher inlet velocity improves the overall air distribution and reduces temperature gradients, it also creates localized regions with excessive air velocity, especially near the occupants located directly below or in front of the air conditioner. In these areas, drafts are created with air velocities of almost 1 m/s near the air conditioner and around 0.5 m/s in other seating areas (150% higher than allowed), causing discomfort and negatively affecting perceived comfort even under acceptable temperature conditions.

Overall, the streamline visualizations demonstrate that an inlet velocity of 1 m/s provides the most balanced air distribution pattern. It ensures sufficient air mixing, minimizes stagnation, and avoids excessive draft, thereby achieving optimal indoor air temperature and velocity distribution and energy efficiency. These findings are consistent with the conclusions drawn from the temperature distribution results.

In the simulations, the k–ε turbulence model was employed to represent the turbulent characteristics of indoor airflow. The turbulent kinetic energy (k) plays an important role in determining the intensity of turbulence and the mixing behavior of the air inside the room. Higher inlet velocities increase the turbulence level near the air conditioner outlet, which enhances air mixing and leads to a more uniform temperature distribution in the indoor environment.

The numerical results obtained in this study are consistent with the findings reported in the literature. Previous CFD studies have demonstrated that increasing inlet air velocity enhances air mixing and leads to a more homogeneous temperature distribution within indoor environments. Similarly, it has been reported that insufficient inlet velocity results in stratification and localized temperature variations, particularly in occupied zones (Othayq, 2025). Also a comparable trend was observed, at an inlet velocity of 0.5 m/s, temperature distribution was non-uniform due to weak air circulation, whereas increasing the velocity improved homogeneity. However, consistent with earlier studies, higher inlet velocities also resulted in increased local air velocities near the diffuser, which may lead to draft risk in occupied regions (Wróblewski et al., 2023).

IV. CONCLUSION

This study investigated indoor air temperature and velocity distribution in an office environment cooled by a split-type air conditioner using CFD. Based on the findings and comparison with international thermal standards (e.g., ASHRAE 55), the following conclusions were reached:

Inlet conditions of 18°C (0.5–1 m/s) and 20°C (1 m/s) were identified as the most effective cases providing homogeneous temperature conditions in the recommended range of 22–26°C.

Higher inlet velocities (2 m/s) resulted in local air velocities exceeding 0.5 m/s while reaching the target temperatures. This represents at least a 150% increase compared to the comfort limit of 0.2 m/s indicating a strong potential for draft-related discomfort.

In the study, it was observed that homogeneous distribution patterns were obtained under specific velocity-temperature rules. It is assumed that such homogeneity can reduce the widening of air conditioner start-stop cycles and may have implications for energy performance. Further research is needed on these expenditures through energy consumption analysis. This study focused on the distribution of air temperature and velocity in the office environment. Future studies will be conducted in more detail, taking into account factors such as relative humidity, the effect of solar radiation, and office occupancy rate.

DECLARATIONS

Acknowledgements: The author do not wish to acknowledge any individual or institution.

Author Contributions: All parts of the study were conducted by the Author.

Conflict of Interest: The author declares no conflict of interest.

Copyright Statement: The author retains the copyright of her work published in the journal, which is licensed under the CC BY-NC 4.0 license, allowing others to share and adapt the work for non-commercial purposes with appropriate attribution.

Supporting Institutions: This research received no external funding.

Ethical Approval: This study does not involve human or animal participants. All procedures followed scientific and ethical principles, and all referenced studies are appropriately cited.

Plagiarism Statement: This article has been evaluated for plagiarism and no instances of plagiarism were detected.

Availability of Data and Materials: Data sharing is not applicable to this study.

Use of AI Tools: The author has employed artificial intelligence (ChatGPT) exclusively for the purpose of acquiring ideas for English sentence structures.

REFERENCES

American Society of Heating, Refrigerating and Air-Conditioning Engineers. (2020). *Thermal environmental conditions for human occupancy* (ANSI/ASHRAE Standard No. 55-2020). ASHRAE.

Awbi, H. B. (2003). Ventilation of buildings (2nd ed.). Taylor & Francis.

Baharudin, N., Manshoor, B., Didane, D. H., Zaman, I., Mazlan, M. A. I., Saddun, A. H. S., Naharazmi, M. N. S., Hasan, M. A. A.,

Sam, M. F. M., Saleman, A. R., & Kumar, S. (2024). Optimization of the air conditioning system for an unoccupied office building using split unit and double-panel radiator. *Journal of Complex Flow, 6*(2), 38–44.

Catalina, T., Virgone, J., & Kuznik, F. (2009). Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling. *Building and Environment, 44*(8), 1740–1750. https://doi.org/10.1016/j.buildenv.2008.11.015

Chen, Q. (2009). Ventilation performance prediction for buildings: A method overview and recent applications. *Building and Environment*, *44*(4), 848–858. https://doi.org/10.1016/j.buildenv.2008.05.025

Fan, M., Fu, Z., Wang, J., Wang, Z., Suo, H., Kong, X., & Li, H. (2022). A review of different ventilation modes on thermal comfort, air quality and virus spread control. *Building and Environment*, *212*, Article 108831. https://doi.org/10.1016/j.buildenv.2022.108831

Fanger, P. O. (1970). *Thermal comfort: Analysis and applications in environmental engineering.* Danish Technical Press.

Ganesh, G. A., Sinha, S. L., Verma, T. N., & Dewangan, S. K. (2024). Energy consumption and thermal comfort assessment using CFD in a naturally ventilated indoor environment under different ventilations. *Thermal Science and Engineering Progress*, *50*, Article 102557. https://doi.org/10.1016/j.tsep.2024.102557

Hatif, I. H., Hariri, A., & Idris, A. F. A. (2020). CFD analysis on effect of air inlet and outlet location on air distribution and thermal comfort in small office. *CFD Letters, 12*(3), 66–77. https://doi.org/10.37934/cfdl.12.3.6677

Horikiri, K., Yao, Y., & Yao, J. (2015). Numerical optimisation of thermal comfort improvement for indoor environment with occupants and furniture. *Energy and Buildings, 88*, 303–315. https://doi.org/10.1016/j.enbuild.2014.12.015

Höfert, O., & Danielak, M. (2021). Comparison of three different ventilation approaches for an open office space regarding pathogen distribution and thermal comfort by CFD. *ASHRAE Transactions, 127*(1), Article VC-21-007.

Huang, Z., Li, Q., He, Y., Ding, X., Dong, Y., & Gao, W. (2025). Coupled heat and mass transfer analysis for indoor air quality and thermal comfort in naturally ventilated offices. *Applied Thermal Engineering, 269*, Article 126019. https://doi.org/10.1016/j.applthermaleng.2025.126019

Lu, Y. C., Chen, C. W., Gong, X. J., Lee, K. R., & Tseng, C. J. (2021). CFD simulations for thermal comfort and energy saving. In *Proceedings of the IEEE/ACIS 22nd International Conference on Software Engineering, Artificial Intelligence, Networking and Parallel/Distributed Computing (SNPD)* (pp. 167–170). IEEE.

Mazlan, M. A. I., Saddun, A. H. S., Naharazmi, M. N. S., Hasan, M. A. A., Sam, M. F. M., Didane, D. H., & Manshoor, B. (2024). CFD simulation of air conditioning system of an office. *Journal of Design for Sustainable and Environment, 6*(2), 48–54.

Mora-Pérez, M., Guillen-Guillamón, I., & López-Jiménez, P. A. (2017). A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions. *AIMS Environmental Science, 4*(2), 289–309. https://doi.org/10.3934/environsci.2017.2.289

Muhamad, S. J., Hamad, H. M., Maheadeen, S. M., & Ahmed, L. R. (2022). An analysis of air flow and temperature distribution using computational fluid dynamics (CFD). *NeuroQuantology, 20*(6), 4932–4942.

Nielsen, P. V. (1998). The selection of turbulence models for prediction of room airflow. ASHRAE Transactions, 104(Part 1B), 1119–1127.

Omerzo, B., Zakula, T., & Glicksman, L. R. (2025). Experimental and CFD study of air distribution system and airflow rate impact on airborne transmission. *Building and Environment*, *271*, Article 112588. https://doi.org/10.1016/j.buildenv.2025.112588

Othayq, M. M. (2025). CFD investigation on the thermal comfort for an office room. *Buildings*, *15*(15), Article 2802. https://doi.org/10.3390/buildings15152802

Pulat, E., & Yuce, B. E. (2021). A parametric study on the effect of blowing parameters of split-type air conditioners on thermal comfort and ventilation. *Heat Transfer Research*, *52*(15), 63–76. https://doi.org/10.1615/HeatTransRes.2021039762

Tang, J., Liu, Y., Du, H., Lan, L., Sun, Y., & Wu, J. (2021). The effects of portable cooling systems on thermal comfort and work performance in a hot environment. *Building Simulation, 14*(6), 1667–1683. https://doi.org/10.1007/s12273-021-0766-y

Ufat, H., Kaynakli, O., Yamankaradeniz, N., & Yamankaradeniz, R. (2017). Three-dimensional air distribution analysis of different outflow typed operating rooms at different inlet velocities and room temperatures. *Advances in Mechanical Engineering, 9*(7), Article 1687814017707414. https://doi.org/10.1177/1687814017707414

Wróblewski, A., Macek, A., Banasiewicz, A., Gola, S., Zawiślak, M., & Janicka, A. (2023). CFD analysis of the forced airflow and temperature distribution in the air-conditioned operator's cabin of the stationary rock breaker in underground mine under increasing heat flux. *Energies, 16*(9), Article 3814. https://doi.org/10.3390/en16093814

Zhang, F., & Ryu, Y. (2021). Simulation study on indoor air distribution and indoor humidity distribution of three ventilation patterns using computational fluid dynamics. *Sustainability*, *13*(7), Article 3630. https://doi.org/10.3390/su13073630

Zhang, Z., Gao, Y., Nie, S., Tian, Y., Li, C., Gao, R., Yin, K., Liu, Y., Liu, B. and & Li, H. (2022). Study on the recommended placement and air distribution of split floor-standing room air conditioners. *International Journal of Architectural Engineering Technology*, *9*, 1–17. https://doi.org/10.15377/2409-9821.2022.09.1