OPEN ACCESS

Volume: 5

Issue: 2

Month: May

Year: 2026

ISSN: 2583-7117

Published: 06.06.2026

Citation:

Bijay Kishore Prasad, Prof. Atul Shanker Suman, "CFD Analysis of Air Conditioning System Enhancement for Thermal Comfort Improvement in Airport Terminals During Summer" International Journal of Innovations in Science Engineering and Management, vol. 5, no. 2, 2026, pp. 394-408.

DOI:

10.69968/ijisem.2026v5i2394-408

CFD Analysis of Air Conditioning System Enhancement for Thermal Comfort Improvement in Airport Terminals During Summer

Bijay Kishore Prasad[1], Prof. Atul Shanker Suman[2]

[1]*Research Scholar, Department of Mechanical Engineering, Corporate Institute of Science and Technology*
[2]*Professor, Department of Mechanical Engineering, Corporate Institute of Science and Technology*

Abstract

The current research examines the thermal environment and thermal comfort of a commercial zone within airport terminals under summer operating conditions through a computational fluid dynamics (CFD) analysis. The main aim of the experiment was to assess the effects of the inlet air temperature, inlet air velocity, and ventilation design on the distribution of the airflow and passenger thermal comfort based on the Chinese national standard GB/T 50785. The commercial zone was modeled as a three dimensional computational model on CATIA V5 and numerically modeled on ANSYS Fluent. The simulations used the Realizable k–ε turbulence model, the SIMPLE pressure–velocity coupling technique, and the second-order upwind discretization scheme. Different modifications in the supply air temperature and inlet airflow velocity were used to test the five cases. The findings revealed that higher inlet airflow velocity and lowering inlet air temperature greatly enhanced airflow circulation and minimized thermal stratification within the commercial area. Case 4 had the highest thermal comfort performance and breathing zone temperatures of nearly the recommended Class I comfort range outlined by GB/T 50785. It is concluded that optimized ventilation parameters can be effectively used to improve indoor thermal comfort and ventilation performance in commercial spaces of airport terminals.

Keywords; Thermal Comfort, Computational Fluid Dynamics (CFD), Airport Terminal Ventilation, Realizable k−ε Model, Indoor Thermal Environment

I. INTRODUCTION

Thermal comfort in the indoor environment has emerged as a research topic because of the growing time that people spend indoors in residential, commercial, educational, and industrial buildings. The indoor environment quality is quite influential on human health, productivity, wellness, and life quality in general [1]. A state of mind known as thermal comfort denotes satisfaction with the thermal environment. Other environmental and personal variables, such as "air temperature, humidity, air velocity, mean radiant temperature, garment insulation, and metabolic activity", also have an impact [2], [3]. A suitable thermal environment must be provided in order to ensure building occupant comfort and energy efficiency. The fast urbanization, climate change, and increasing energy efficiency requirement in buildings have heightened the necessity of effective thermal comfort management strategies [4]. "Heating, ventilation and air conditioning (HVAC) systems" are common in present-day buildings to ensure that the required conditions are maintained indoors. But, mis-design or functioning of these systems can result in uneven temperature distribution, inadequate circulation of air, high energy use and occupant discomfort. Thus, the thermal performance of indoor environments is now a critical issue to learn how to design comfortable and sustainable living and working areas [5], [6]

Thermal comfort is the subject of study that examines the airflow patterns, heat transfer process, temperature distribution, and ventilation efficiency in enclosed spaces. Indoor thermal conditions and ventilation system performance are usually determined by use of "Computational Fluid Dynamics (CFD)" and experimental methods [7], [8]. The positioning of air inlets and outlets, airflow velocity, occupancy level, and seasonal variations are factors that are important in the determination of conditions of indoor comfort. To improve indoor air quality, reduce thermal stress, and reduce energy consumption, these parameters may be effectively adjusted [9], [10]. Over the past years, scientists have been working on the expansion of new technologies that can help to improve thermal comfort, minimizing the impact on the environment and the associated costs [11], [12]. There is growing use of sustainable building technologies, energy efficient HVAC systems, passive cooling, and smart ventilation strategies to attain improved indoor environmental conditions. The combination of renewable energy systems and smart control technologies has also helped to enhance the indoor thermal performance [13]..

Importance of Thermal Comfort in Airport Terminals

One of the critical areas of indoor environmental quality in airport terminals is thermal comfort since these buildings host a significant number of passengers, visitors, and employees over a prolonged period. Airport terminals are an intricate interior setting which is densely populated, operates 24 hours and has a fluctuating climate [14], [15]. To maintain customer satisfaction, employee productivity and general operational efficiency, there is need to maintain appropriate thermal conditions within such spaces. Thermal comfort is a condition where the occupants are satisfied with the surrounding environment thermally [16]. Factors affecting it include air temperature, humidity, airflow, radiant heat and human activity. Airport terminals are built to accommodate people who have various climatic conditions and physical abilities; hence, it becomes difficult to ensure that there are similar thermal conditions [17], [18]. Long queues, overcrowded space, or uneven distribution of temperature may cause discomfort to the passengers. Poor thermal conditions may lead to stress levels, fatigue and dissatisfaction among passengers. Likewise, when employees spend long periods of time in unfavorable thermal conditions, they can lose concentration, productivity, and health issues [19], [20].

There is also a direct correlation between indoor air quality and energy efficiency in thermal comfort. Adequate ventilation and airflow ensures fresh air flow and minimizes the build up of pollutants and odors. The HVAC (heating, ventilation, and air conditioning) systems of airport terminals consume a lot of energy [21], [22]. Thus, energy-saving and the accomplishment of thermal comfort have become a significant goal in sustainable building design. The development of HVAC technologies, intelligent control systems, and computational analysis techniques has enhanced the capability to control and optimize the thermal requirements in the indoor settings [23], [24]. Scholars are paying more attention to airflow distribution, minimizing thermal stratification, and maximizing energy efficiency within airport buildings. Adequate management of thermal comfort is not only good in enhancing the well-being of occupants, but also in ensuring sustainable development and environmental conservation. Thus, research on thermal comfort in airport terminals is vital in creating healthier, more efficient, and energy-conscious indoor spaces [25], [26]

Role of HVAC Systems in Thermal Comfort

The Heating, Ventilation and Air Conditioning (HVAC) systems are also important in ensuring thermal comfort and air quality in the airport terminals and other big indoor areas. HVAC systems are meant to control the temperature, humidity, air movement and the purity of air in the inside environment so as to provide comfortable conditions to the occupants [27]. Airport terminals are busy 24/7, and they serve many people; there is a need to ensure that the HVACs are running effectively to comfort passengers and save on energy. The heating part of HVAC systems makes the appropriate temperatures at the interiors in case of cold weather, and the cooling part removes unnecessary heat in times of summer [28], [29]. The ventilation systems bring fresh air outside and take away contaminated indoor air, hence, enhancing the indoor air quality and decreasing the levels of pollutants. When properly ventilated, they also aid in controlling the level of the humidity and avoid the build-up of odours, bacteria and air contaminants [30], [31].

Another important role of HVAC systems is air distribution. The supply diffusers and return outlets should be properly placed to have even distribution of air flow and temperature in the inside environments. Lack of uniformity in air circulation can cause hot and cold spots, which can cause occupants to feel uncomfortable [32]. Advanced HVAC systems employ sensors, automated controls, and smart technologies to constantly measure the conditions within the indoor environment and respond to system performance. The energy consumption of airport buildings is also significantly influenced by HVAC systems [33]. The demanding energy requirements are due to large cooling and heating loads, high occupancy density, and extensive operating hours. Thus, the HVAC systems that are more energy-efficient (e.g. the use of variable air volume systems, energy recovery ventilation, smart thermostats, integration of renewable energy) are becoming more popular to enhance the efficiency of the systems and make them cheaper to operate [34]. HVAC systems directly affect the thermal comfort, occupant satisfaction, and building sustainability. Design, maintenance, and optimization of HVAC systems are useful to achieve the stable indoor thermal condition and reduce the amount of energy used and the environmental impact. The HVAC systems are therefore regarded to be the foundation of the indoor environmental control in contemporary airport terminals and enormous commercial buildings [35].

Objective

1. To study the thermal comfort inside the airport terminals in the summer season.
2. To study the effect of variation in the inlet and outlet vent position in the commercial space of the airport terminals.
3. To study the effect of variation in the temperature of the inlet vent in thermal comfort.
4. To study the effect of variation in the velocity of the inlet air in thermal comfort.
5. To study evaluate the thermal comfort in the commercial space of the airport terminals according to the General Chinese Standards (GB/T 50785).

RESEARCH METHODOLOGY

Geometry Description

The computational domain used in the present study was designed to investigate the airflow distribution, ventilation performance, and thermal environment inside a commercial island space. The geometric model of the commercial island had overall dimensions of 13 m in length, 9 m in width, and 4.2 m in height. To accurately capture the surrounding airflow behavior and minimize boundary effects during “the Computational Fluid Dynamics (CFD) simulation”, a computational domain of commercial zone was developed with dimensions of 39.5 m × 19 m × 9 m.

The ventilation system configuration consisted of two different size of inlet vents positioned near the ceiling level on the walls of the commercial island. There are 5 inlet vent are placed in the left and right side wall, and 6 inlet vent are placed in the back side wall. The first size of inlet vent had dimensions of 0.6 m × 0.15 m, while the second size of inlet vent measured 0.65 m × 0.2 m. In order to distribute airflow evenly throughout the interior space, the smaller inlet vents were symmetrically placed on the commercial island's left and right side walls. The positioning of the inlet vents near the ceiling was intended to enhance air circulation and improve ventilation effectiveness throughout the occupied zone

An outlet vent with dimensions of 1.6 m × 0.6 m was installed on the front wall of the commercial island. The outlet vent was designed to facilitate the removal of indoor air and maintain continuous airflow circulation within the computational domain. The arrangement of inlet and outlet vents was selected to investigate the influence of ventilation configuration on “airflow characteristics, temperature distribution, and thermal comfort conditions”.

For simplification of the computational model, seated passengers and seating arrangements were represented as a simplified rectangular block with dimensions of 3.4 m × 1.2 m × 1.2 m. This simplification reduced computational complexity while preserving the influence of occupant obstruction on airflow patterns and heat transfer behavior. The developed geometric model was subsequently used for mesh generation and numerical analysis in ANSYS Fluent to evaluate indoor thermal and ventilation performance.

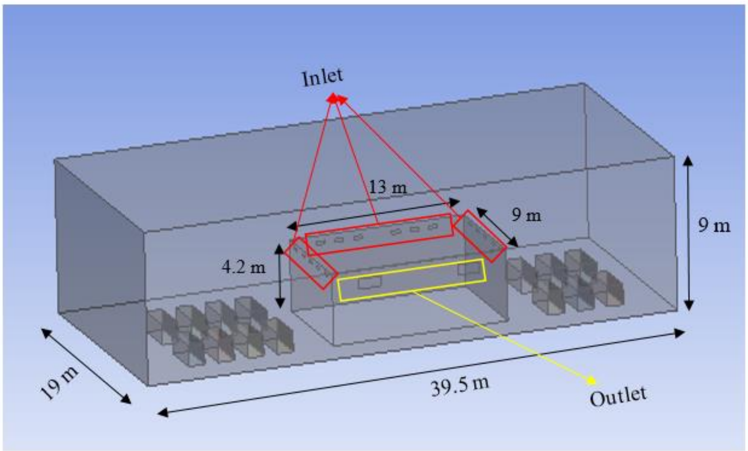
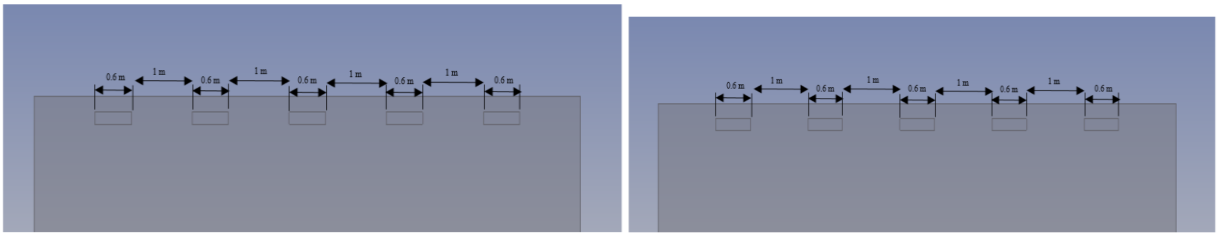
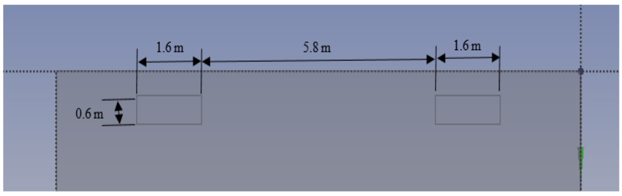


Figure 1: Computational domain of Commercial Island



Inlet vent at left and right side wall

Inlet vent at back wall



Outlet vent at front wall

Figure 2: Inlet and outlet vent dimension

Chinese national standard GB/T 50785

The Chinese national standard GB/T 50785 named Evaluation Standard for Indoor Thermal Environment in Civil Buildings gives guidelines for evaluation of thermal comfort conditions in indoor environments during summer and winter seasons. This standard is widely used in China for evaluating indoor environmental quality in residential, commercial, institutional, and public buildings. The standard establishes acceptable indoor thermal parameters to ensure occupant comfort, health, and energy-efficient building operation. GB/T 50785 evaluates indoor thermal comfort by considering environmental factors including "indoor air temperature, relative humidity, air velocity and thermal sensation". The standard classifies indoor thermal environments into different comfort categories according to occupant satisfaction levels. The parameters were registered at 1.1 m, height that better represents the environmental comfort. Among these, Class I and Class II thermal comfort conditions are commonly adopted for building design and environmental assessment

Table 1: Class I and Class II thermal comfort conditions

	Summer Season Conditions	**Winter Season Conditions**
Class I Thermal Comfort Conditions		
Indoor operative temperature	24°C – 26°C	20°C – 24°C
Relative humidity	40% – 60%	30% – 60%
Air velocity	0.25 m/s	≤ 0.20 m/s
Class II Thermal Comfort Conditions		
Indoor operative temperature	26°C – 28°C	18°C – 20°C
Relative humidity	30% – 70%	30% – 70%
Air velocity	≤ 0.30 m/s	≤ 0.25 m/s

Compared with Class I, Class II conditions allow wider thermal parameter ranges and slightly lower thermal comfort expectations while reducing energy consumption. The GB/T 50785 standard is important for evaluating indoor thermal environments and HVAC system performance in building engineering studies. The standard provides reference values of temperature distribution, humidity levels, airflow velocity and thermal comfort of occupants for "computational fluid dynamics (CFD) simulations and experimental investigations". The standard also supports sustainable building design by balancing occupant comfort with energy efficiency. By following GB/T 50785 guidelines, designers and researchers can "optimize ventilation systems, improve indoor air quality, and reduce energy consumption" while maintaining acceptable thermal conditions in indoor environments.

Numerical simulation

Based on the measured data and specified initial boundary conditions, numerical simulations were conducted using ANSYS Fluent software to investigate the indoor thermal environment and airflow characteristics within the commercial zone. The simulations were performed under steady-state conditions to evaluate airflow distribution, temperature variation, and thermal comfort performance inside the computational domain. In the present study, "the

Realizable k−ε turbulence model" was employed for turbulence modeling. This model was selected because of its improved capability to accurately predict complex turbulent flow behavior, including jet flows, recirculation regions, impinging flows, rotating flows, vortex structures, and thermally buoyant airflow. Furthermore, "the Realizable k−ε model" has been widely applied in indoor environmental and ventilation studies due to its reliable prediction of airflow and thermal characteristics in enclosed spaces. The pressure–velocity coupling during the numerical solution process was achieved using "the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm". The SIMPLE algorithm is extensively utilized in "Computational Fluid Dynamics (CFD) simulations" for indoor airflow analysis because of its numerical stability, convergence efficiency, and suitability for incompressible flow conditions commonly encountered in built environments.

For spatial discretization of the governing equations, "the second-order upwind scheme was adopted for the momentum, energy, turbulent kinetic energy (k), and turbulent dissipation rate (ε) equations". The second-order discretization method improves numerical accuracy by reducing discretization errors and enhancing solution stability for complex airflow and heat transfer simulations. The standard wall function approach was used in the near-wall regions to accurately model near-wall turbulent flow behavior. The use of wall functions reduces computational cost while minimizing inaccuracies associated with turbulence modeling near solid boundaries. This approach is commonly used in indoor airflow simulations where wall-bounded turbulent flows are dominant. The convergence criteria for the numerical simulations were carefully defined to ensure solution accuracy and reliability. Residual convergence limits were set to 10^{-6} for the energy, "continuity, momentum, turbulent kinetic energy (k), and turbulent dissipation rate (ε) equations". These convergence settings ensured stable numerical solutions and reliable prediction of indoor thermal environmental conditions.

Governing equation

In "Computational Fluid Dynamics (CFD) analysis using ANSYS Fluent", the flow behavior, heat transfer, and turbulence characteristics are governed by fundamental conservation equations. In the present study, the Realizable k−ε turbulence model is used along with the SIMPLE algorithm for pressure–velocity coupling. These equations are solved numerically using the finite volume method to analyze airflow, temperature distribution, and thermal comfort inside indoor spaces.

Continuity Equation

The continuity equation ensures that mass is conserved throughout the computational domain. For indoor airflow analysis, incompressible flow conditions are generally assumed because air velocity is relatively low. The continuity equation represents the conservation of mass in fluid flow.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0$$

For incompressible steady flow:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

Momentum Conservation Equation

This equation governs airflow motion due to pressure gradients, viscous forces, and gravity effects. It is essential for predicting airflow distribution and ventilation characteristics inside indoor environments. The momentum equation is based on Newton's second law and describes fluid motion.

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i$$

Turbulent Kinetic Energy Equation (k)

This equation calculates turbulent kinetic energy generated due to velocity fluctuations. It helps model turbulent airflow behavior in indoor spaces. The turbulent kinetic energy equation in the Realizable k−ε model is:

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

Energy Dissipation Rate Equation (ε)

This equation determines the rate at which turbulent kinetic energy dissipates into thermal energy due to viscous effects. The dissipation rate equation for the Realizable k−ε model is:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+\rho C_1 S\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}$$

Energy Conservation Equation

The energy equation predicts temperature distribution and heat transfer inside indoor environments. It is essential for thermal comfort analysis. The energy equation governs heat transfer within the fluid domain.

$$\frac{\partial(\rho E)}{\partial t}+\frac{\partial[u_i(\rho E+p)]}{\partial x_i}=\frac{\partial}{\partial x_j}\left(k_{eff}\frac{\partial T}{\partial x_j}\right)+S_h$$

Boundary condition

In this numerical study, suitable boundary conditions were imposed in ANSYS Fluent in order to properly simulate the indoor airflow and thermal environment of the commercial zone. The velocity inlet boundary condition was used to model the air supply vents, with inlet airflow velocity ranging from 3.5 m/s to 6.5 m/s to investigate the effect of airflow rate on the indoor thermal comfort and ventilation performance. Inlet air temperature was defined to be between 19 °C and 21 °C, depending on the operating conditions considered in the present study. The pressure outlet boundary conditions were set as reference pressure in the atmosphere to remove the indoor air from the computational domain, while the return air vents were set as pressure outlet boundary conditions. The configuration provided a steady flow circulation and kept the mass conservation constant in the numerical simulation. The walls, floor, and roof surfaces of the commercial zone were modeled as stationary no-slip wall boundary conditions. These solid surfaces were treated with fixed wall temperature conditions to consider heat transfer interactions between the indoor air and surrounding solid surfaces. This non-slip condition allows the fluid velocity at the wall surfaces to be zero, which is a good representation of the near wall flow behavior. The thermal effect from passengers seated in the cabin was simulated by placing the human body with the seating arrangement in the model to represent a constant heat flux source. The metabolic heat produced in an active seated condition by the occupants was modeled using a uniform heat flux boundary condition of 69.78 W/m². The boundary conditions were set up to simulate the airflow distribution, thermal variation and thermal comfort properties inside the building accurately.

Table 2: Inlet condition in all cases

Cases		Inlet at left side wall	Inlet at back side wall	Inlet right side wall
Case 1	Temperature (°C)	21	21	21
	Velocity (m/s)	3.5	3.5	3.5
Case 2	Temperature (°C)	21	21	21
	Velocity (m/s)	4	4	4
Case 3	Temperature (°C)	20	20	20
	Velocity (m/s)	4	4	4
Case 4	Temperature (°C)	20	20	20
	Velocity (m/s)	5	5	5
Case 5	Temperature (°C)	19	19	20
	Velocity (m/s)	6.5	6.5	6

Measurement zone and test points

In the present study, five different simulation cases were considered in which the inlet air temperature and inlet air velocity were varied according to the operating conditions specified in the corresponding table. To examine the effects of the ventilation parameters on intra-commercial zone air flow distribution, temperature changes, and thermal comfort, these cases were developed. The number of monitoring planes and measuring points in the computational domain were set to evaluate the thermal environment inside and the comfort conditions of the occupants. Two monitoring points were selected within the occupied zone. The first point was located at the center of the seated passenger region on the left side of the commercial zone, while the second point was positioned at the center of the seated passenger region on the right side. Both measurement points were located at a height of 1.1 m above the floor level, corresponding to the breathing zone of seated occupants. Additionally, four measurement planes were considered for temperature contour analysis. One horizontal plane was positioned at 1.1 m above the floor surface, while three vertical planes were located at distances of 4.5 m, 7.5 m, and 9.5 m from the front wall of the commercial zone. These monitoring planes and measurement locations are illustrated in the corresponding figure.

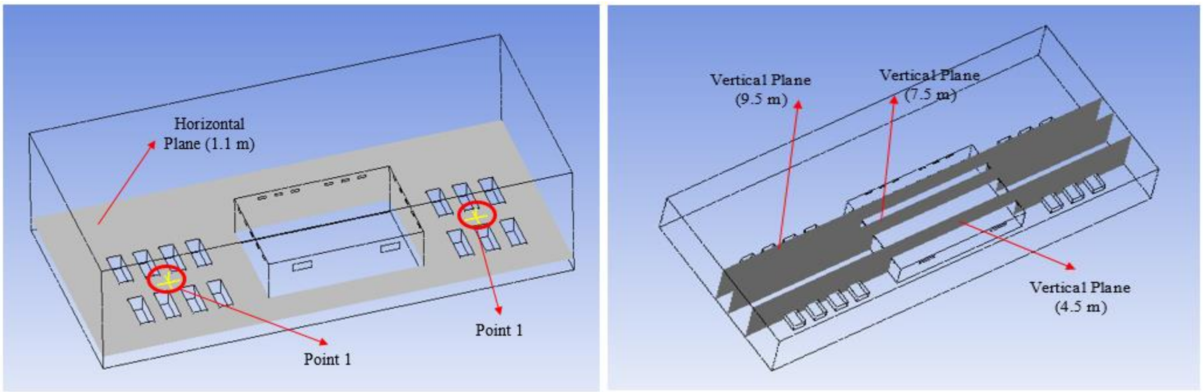
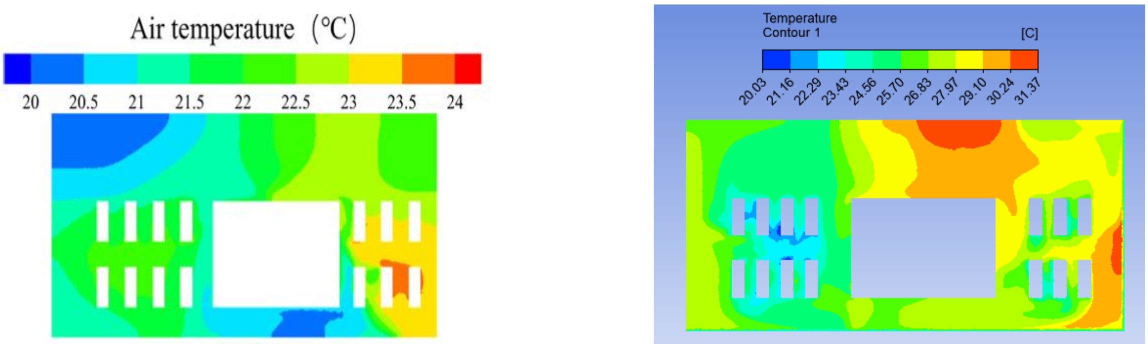


Figure 3: Measurement planes and points

Model validation

To validate the numerical model and boundary conditions adopted in the present study, “Computational Fluid Dynamics (CFD) simulations” were performed using “ANSYS Fluent software”, and the predicted thermal parameters were compared with previously published experimental and numerical results reported by Pengyu Yan et al. (2026). The validation process was conducted to ensure the reliability and accuracy of the developed computational model for predicting indoor thermal comfort and airflow behavior within the commercial zone. The computational domain was developed with overall dimensions of 39.5 m × 19 m × 9 m to adequately capture airflow circulation and minimize boundary interference effects. The commercial island located inside the computational domain had dimensions of 13 m × 9 m × 4.2 m. A total of fourteen air supply vents were incorporated into the geometric model. Among these, inlet vents 1–4 were positioned on the left-side wall, inlet vents 5–10 were arranged on the back-side wall, and inlet vents 11–14 were installed on the right-side wall of the commercial island. Each inlet vent had dimensions of 0.65 m × 0.15 m, while the outlet vent dimensions were 1.6 m × 0.6 m.

The numerical simulations were performed by the Realizable k−ε model, which was known as their favorite model because it was good at predicting the features of turbulent airflow and ventilation in an indoor environment. The SIMPLE algorithm was employed to achieve pressure–velocity coupling, and the governing equations of continuity, momentum, energy, turbulent kinetic energy (k) and turbulent dissipation rate (ε) were discretized by the second-order upwind scheme for better numerical stability and solution accuracy. All the governing equations were resolved numerically with the convergence criteria of 10−6 to assure the accuracy and stability of the numerical solution. The validation was carried out by comparing the air temperature values obtained from the simulation at the breathing zone level, which is 1.1 m away from the floor surface near the left side of the commercial island where the seated passengers are located. The inlet air velocities and temperatures of the vents (1-4) and (5-10) were set based on the measured sampling information that was collected at the breathing zone locations. The comparison between the simulated results and the reference study demonstrated excellent agreement, with the relative error remaining below 1.8%. The low deviation confirmed that the developed computational domain, turbulence model, and boundary condition settings were reliable and suitable for predicting indoor airflow distribution, temperature variation, and thermal comfort conditions within the commercial zone.



Temperature contour of Pengyu Yan et al. (2026) Temperature contour of present simulation

Figure 4: Temperature contour validation

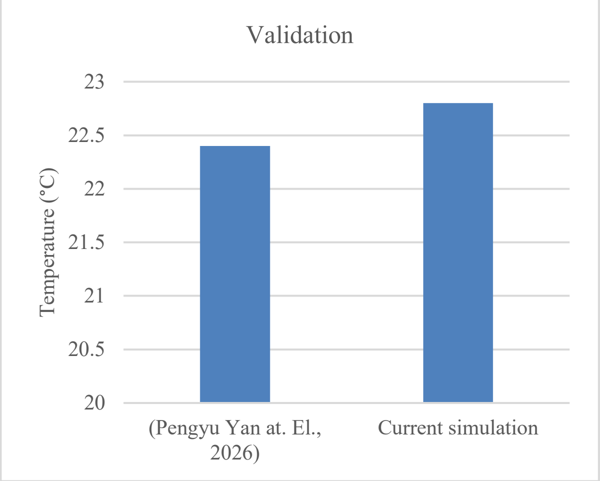


Figure 5: Temperature validate of present simulation from Pengyu Yan et al. (2026)

RESULT AND DISCUSSION

This section discusses the thermal comfort performance within the commercial zone based on temperature contour distribution and point-wise temperature analysis for all simulated cases. The temperature contours were evaluated on four different sectional planes to investigate the spatial temperature distribution and airflow behavior throughout the computational domain. Thermal comfort conditions were assessed according to "the Chinese national standard GB/T 50785 for indoor thermal environment evaluation". Initially, the modified computational model of the commercial zone was validated under summer operating conditions using the base design reported by Pengyu Yan et al. (2026). The base design refers to the original ventilation configuration and operating conditions adopted in the reference study. The validation process verified the reliability and accuracy of the developed numerical model for predicting the indoor airflow and thermal characteristics. Following the validation stage, parametric investigations were conducted by varying the inlet air temperature and inlet air velocity to analyze their influence on indoor thermal comfort, temperature distribution, and ventilation effectiveness within the commercial zone.

Comparison of modified design with base design

The original study conducted by Pengyu Yan et al. (2026) primarily focused on the thermal environment of the commercial zone under winter operating conditions. In the present investigation, the geometry of the commercial island was modified to evaluate indoor thermal comfort performance during summer conditions. Initially, the base design proposed by Pengyu Yan et al. (2026) was numerically simulated under summer operating conditions to establish a reference case. Subsequently, the modified commercial zone configuration was analyzed and validated through comparative thermal assessment. For the validation study, the inlet air temperature and inlet air velocity were maintained at 21 °C and 3.5 m/s, respectively, for both computational models. Figure illustrates the comparison of temperature distributions obtained from the base design and the modified design at selected monitoring points within the occupied zone. At Point 1, the predicted temperature for the base design was 28.28 °C, whereas the modified design exhibited a reduced temperature of 27.25 °C. The relative temperature reduction between the two configurations was approximately 3.64%. At Point 2, the simulated temperature values for the base design and modified design were 26.25 °C and 26.34 °C, respectively, corresponding to a minor variation of 0.34%. The results indicate that the modified commercial zone configuration achieved improved thermal conditions at Point 1 by reducing the local indoor temperature within the occupied region. Since the modified design demonstrated comparatively better thermal comfort performance under summer conditions, it was selected for further parametric investigation involving variations in inlet air temperature and airflow velocity.

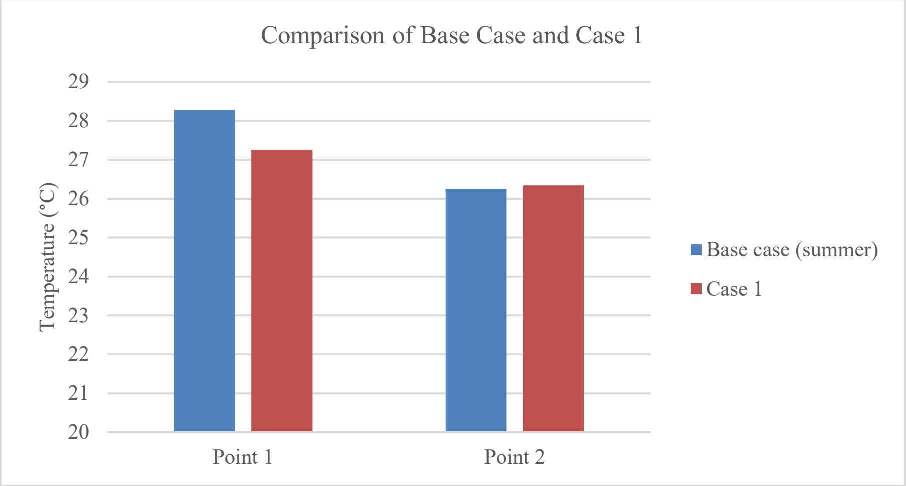


Figure 6: Comparison of base case and case 1

Temperature contour

The temperature distribution analysis is presented for all five simulated cases at the horizontal plane 1.1 m above the floor level as well as at three vertical planes at 4.5 m, 7.5 m and 9.5 m inside the computational domain. The horizontal plane at 1.1 m is the breathing zone height of seated passengers and this level is significant for thermal comfort assessment of occupants. The thermal performance of each case was also evaluated by taking temperature measurements at Point 1 and Point 2, which are near the left and right sides of the seated passenger compartment, respectively.

The simulation results indicate a continuous reduction in both maximum and average temperatures from Case 1 to Case 5 at all measurement planes. At the horizontal plane, the maximum temperatures decreased from 37.368 °C in Case 1 to 32.00 °C in Case 5, while the average temperatures reduced from 25.61 °C to 23.11 °C. This reduction occurred due to the increase in inlet air velocity and decrease in inlet air temperature, which enhanced convective heat transfer and improved airflow circulation inside the commercial zone. Higher inlet velocities promoted more effective mixing between conditioned air and indoor warm air, thereby reducing localized heat accumulation.

At the vertical plane located at 4.5 m, the maximum temperatures gradually reduced from 33.04 °C in Case 1 to 32.00 °C in Case 5, while the average temperatures decreased from 24.59 °C to 22.34 °C. Similar behavior was observed at the 7.5 m vertical plane, where the maximum temperature remained constant at 32.00 °C for all cases; however, the average temperature progressively reduced from 24.86 °C to 22.52 °C. The comparatively stable maximum temperature at this plane indicates that the upper airflow region experienced more uniform thermal conditions due to effective air mixing and ventilation circulation.

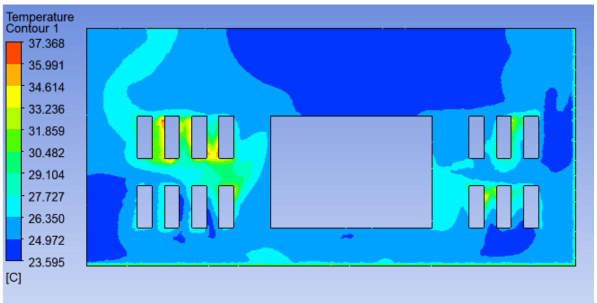
The highest temperature variations were observed at the vertical plane located at 9.5 m. The maximum temperature reduced significantly from 48.69 °C in Case 1 to 35.63 °C in Case 5, while the average temperature decreased from 24.78 °C to 22.46 °C. This substantial reduction demonstrates that increasing inlet airflow velocity effectively minimized thermal stratification and prevented excessive heat accumulation near the upper region of the commercial zone. In the lower airflow cases, warm air tended to rise and remain trapped near the ceiling due to buoyancy effects, resulting in higher upper-zone temperatures.

The temperature difference at Point 1 and Point 2 also showed an improvement in the thermal comfort conditions from Case 1 to Case 5. At Point 1, the temperature reduced from 27.25 °C to 23.42 °C, while at Point 2, the temperature decreased from 26.34 °C to 23.84 °C. The drop in local temperatures is due to the better cooling capabilities and the performance of the ventilation that occurs in high airflow. Case 5 was the best thermal comfort performance because it had the lowest inlet temperature and highest inlet air velocity, which resulted in better penetration and less non-uniformity of temperature in the occupied zone.

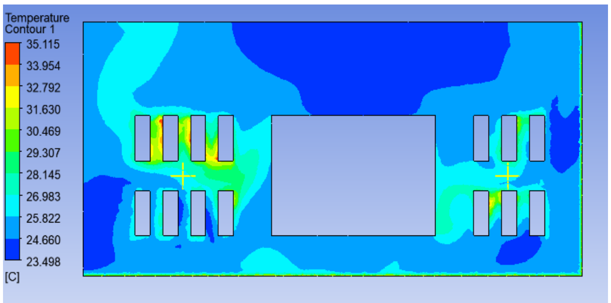
Overall, the results demonstrate that increasing inlet air velocity and reducing supply air temperature significantly improve indoor thermal conditions by enhancing airflow distribution, reducing thermal stratification, and improving convective heat removal within the commercial zone.

Table 3: Result in all cases at points and planes

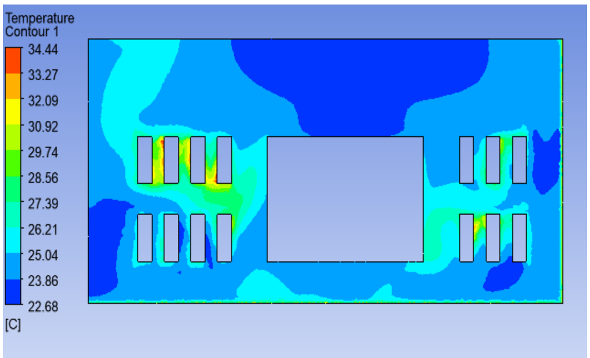
	Case 1	Case 2	Case 3	Case 4	Case 5
Point 1	27.25	26.40	25.62	25.30	23.42
Point 2	26.34	26.09	25.37	24.88	23.84
Horizontal plane (1.1 m)	25.61	25.36	24.63	24.28	23.11
Vertical plane (4.5 m)	24.59	24.44	23.66	23.42	22.34
Vertical plane (7.5 m)	24.86	24.76	23.89	23.61	22.52
Vertical plane (9.5 m)	24.78	24.62	23.85	23.56	22.46



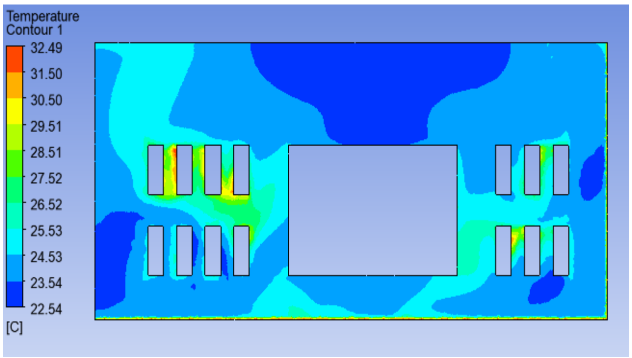
Case 1



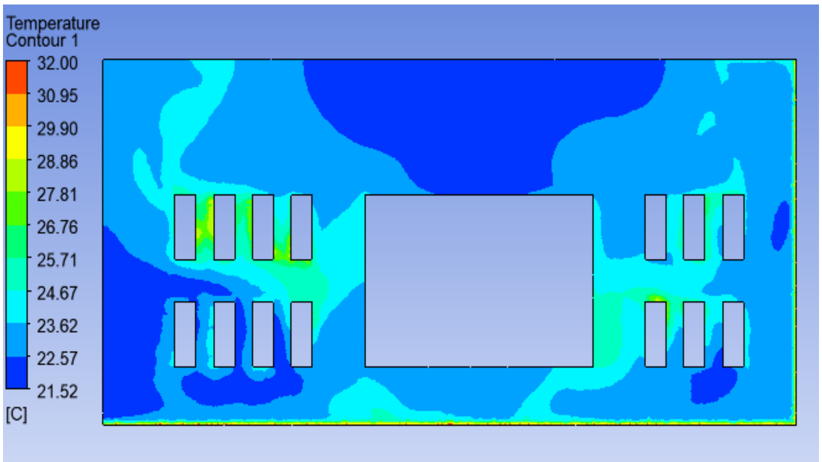
Case 2



Case 3

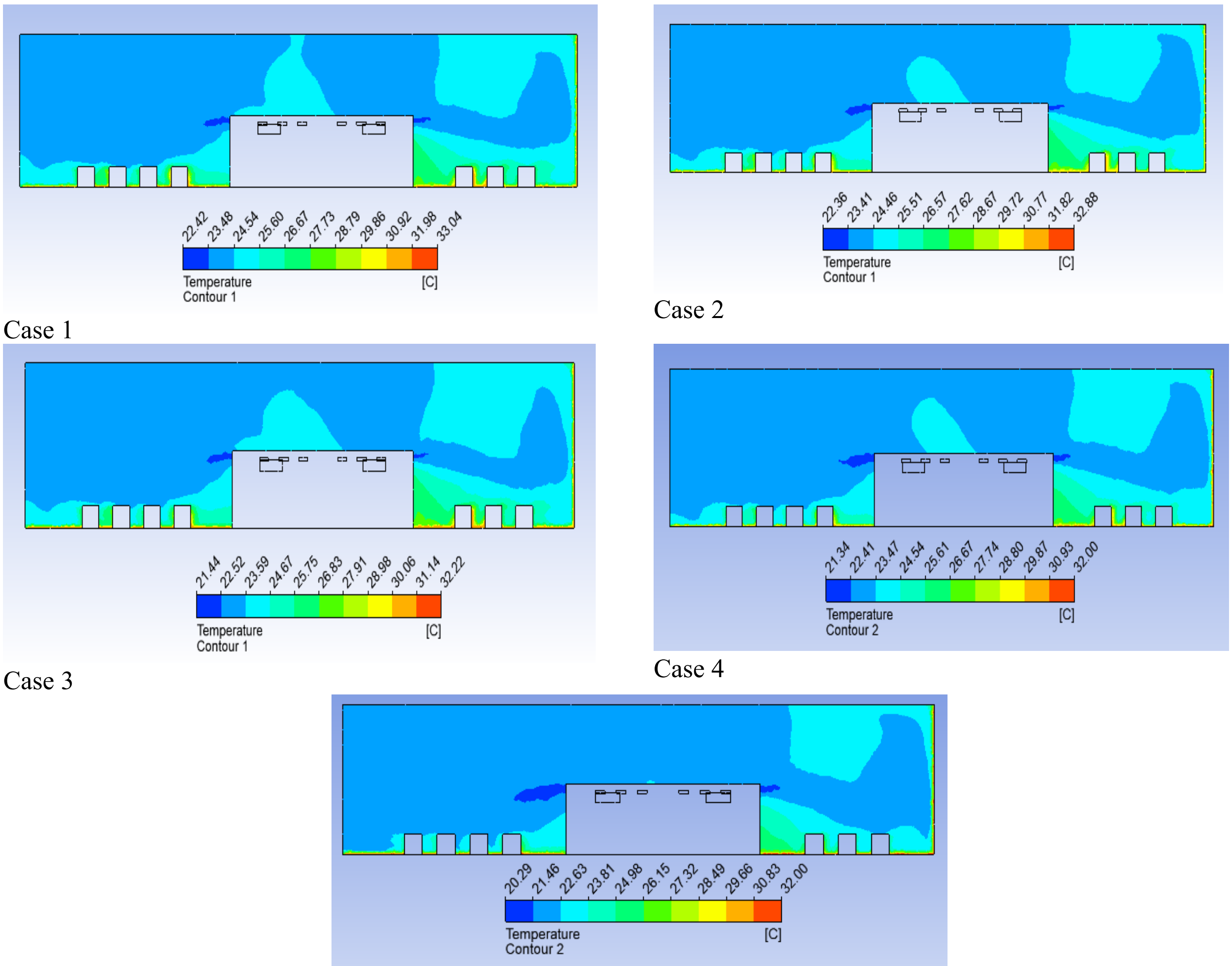


Case 4



Case 5

Figure 7: Temperature contour at horizontal plane (1.1 m) in all cases



Case 1

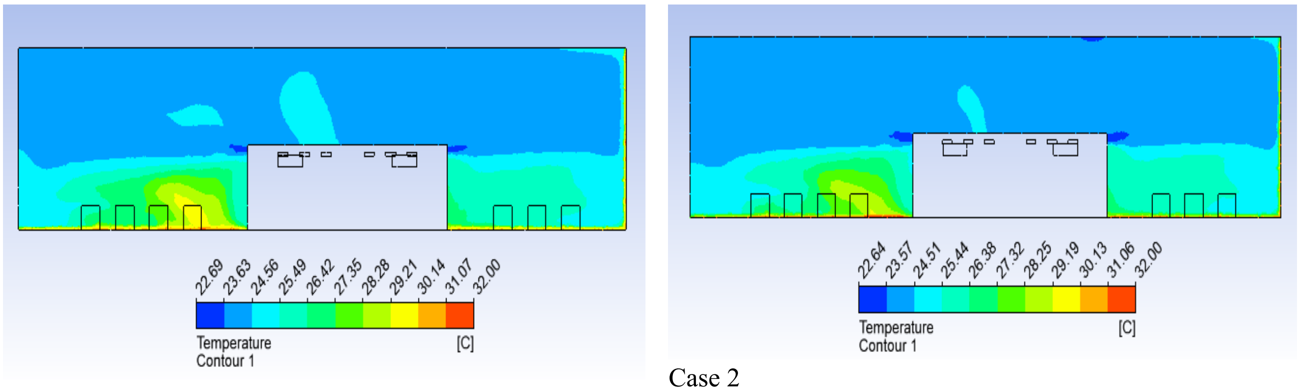
Case 2

Case 3

Case 4

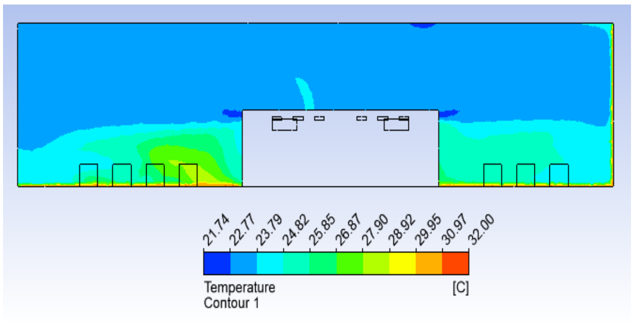
Case 5

Figure 8: Temperature contour at vertical plane (4.5 m) in all cases

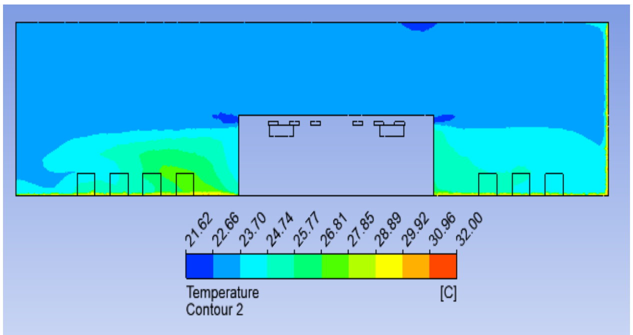


Case 1

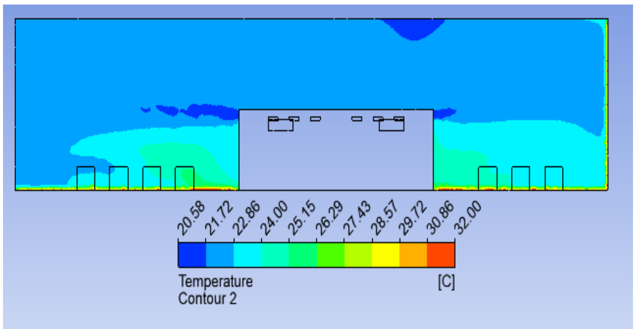
Case 2



Case 3

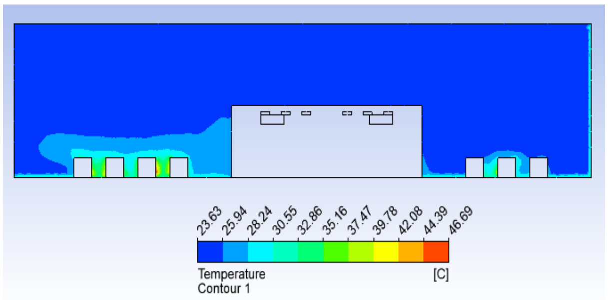


Case 4

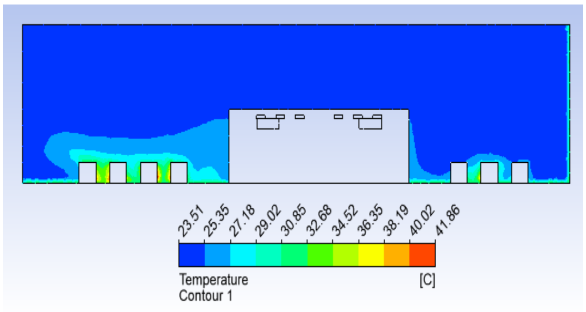


Case 5

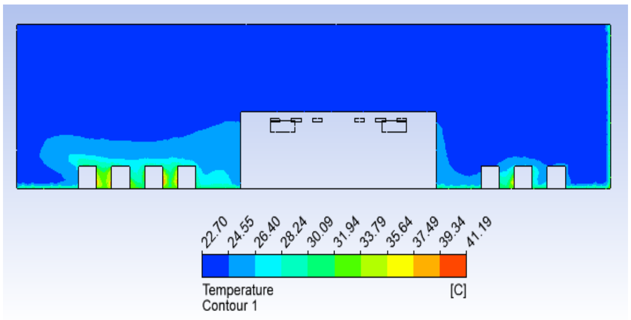
Figure 9: Temperature contour at vertical plane (7.5 m) in all cases



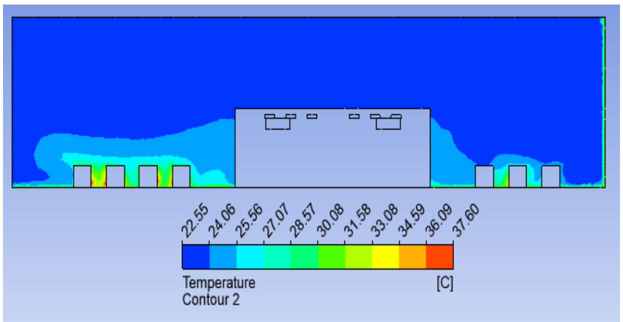
Case 1



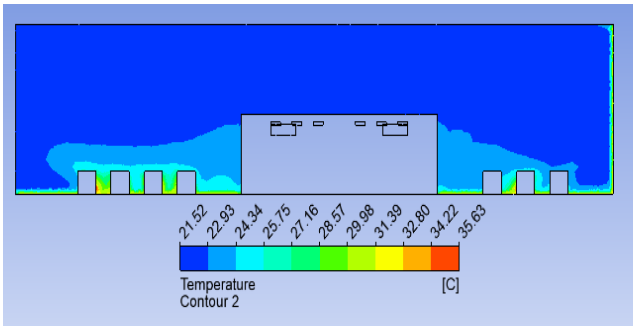
Case 2



Case 3



Case 4



Case 5

Figure 10: Temperature contour at vertical plane (9.5 m) in all cases

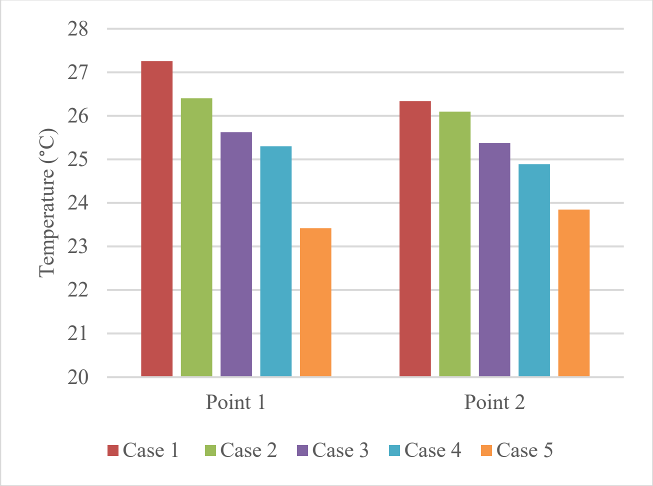


Figure 11: temperature of all cases

CONCLUSION

The current research examined the thermal environment and thermal comfort of a commercial zone under the summer condition of operation through “a Computational Fluid Dynamics (CFD) analysis of ANSYS Fluent”. Five cases were modeled by changing inlet air temperature, and inlet air velocity to assess their effect on airflow distribution, temperature change, and passenger thermal comfort in the occupied zone. The “Realizable k−ε turbulence model and SIMPLE pressure–velocity coupling algorithm” were successfully implemented to predict the indoor airflow and thermal characteristics. The outcomes of the simulation indicated that maximum and average temperatures steadily reduced as Case 1 to Case 5 in all the monitoring planes and occupant breathing zones. According to the Chinese national standard GB/T 50785, thermal comfort study revealed that although Case 3, Case 4, and Case 5 inclined toward "Class I thermal comfort conditions in the occupied zone", Cases 1 and 2 tended toward Class II thermal comfort conditions.

1. The reduction in temperature occurred due to improved airflow circulation, enhanced convective heat transfer, and reduced thermal stratification caused by higher inlet airflow velocity and lower supply air temperature.
2. Significant temperature reduction was observed particularly at the upper vertical plane (9.5 m), indicating that the optimized ventilation conditions effectively minimized heat accumulation near the ceiling region.
3. Among all simulated cases, Case 4 provided the most favorable thermal environment, with breathing zone temperatures of 25.30 °C and 24.88 °C at Point 1 and Point 2, respectively. These values were very close to the recommended Class I summer comfort range of 24 °C–26 °C specified in GB/T 50785.

Overall, the study discovered that the commercial zone's "thermal comfort and ventilation efficiency" are significantly improved by increasing the input airflow velocity and decreasing the supply air temperature. The optimized ventilation layout effectively increased distribution of air flow, localized heat, and thermal comfort of occupants during summer operating conditions. Thus, the results of the paper can be used to design and optimize energy efficient HVAC systems and sustainable indoor environment management of commercial buildings and terminal spaces at airports.

REFERENCES

[1] M. Wang, H. Zhang, J. Zhang, and J. Ao, “A Study on Summer Thermal Comfort in Chongqing Riverside Parks: Based on Microclimate Measurements and Thermal Comfort Evaluation,” *Sustainability*, vol. 18, no. 4990, 2026.

[2] A. MOHAN, V. R, and R. DEIVENDIRAN, “THERMAL COMFORT ANALYSIS IN AN EDUCATIONAL BUILDING IN A HOT CLIMATE AREA USING CFD,” *Therm. Sci.*, vol. 30, no. 1, pp. 275–285, 2026.

[3] H. S. Malik, W. Khalid, A. Jabbar, A. Munir, and A. Waqas, "Comparative Analysis of Thermal Comfort Surveys with CFD Simulations," *Master Conf. People Build.*, 2022.

[4] L. Jiang *et al.*, "Thermal Environment and Thermal Comfort of Modern Timber Buildings: A Systematic Review," *Buildings*, vol. 16, no. 1966, pp. 1–38, 2026.

[5] A. Victoria, P. Florido, M. M. Ouf, W. O. Brien, N. Cooper, and H. Awad, "Thermal Comfort and Passenger Responses in Airports," *E3S Web Conf.*, vol. 689, no. 06007, pp. 1–7, 2026.

[6] T. Lim and D. D. Kim, "Thermal Comfort Assessment of the Perimeter Zones by Using CFD Simulation," *Sustainability*, vol. 14, 2022, doi: 10.3390/su142315647.

[7] N. B. Chien, V. T. Ngoc, N. D. Vinh, T. M. Thang, and H. H. Phung, "CFD Simulation Analysis of Thermal Comfort in a Small Office," *Evergr. - Jt. J. Nov. Carbon Resour. Sci. Green Asia Strateg.*, vol. 12, no. 04, pp. 2216–2222, 2025.

[8] A. Benabed and A. Boulbair, "Numerical analysis of thermal comfort and air freshness generated by a multi-cone diffuser with and without lobed inserts," *J. Build. Eng.*, vol. 54, 2022.

[9] X. Wang and Y. Pan, "Utilizing computational fluid dynamics (CFD) for simulating airflow and heat distribution to enhance thermal comfort in enclosed space," *J. Phys. Conf. Ser.*, 2025, doi: 10.1088/1742-6596/2949/1/012049.

[10] Y. H. Yau, H. S. Toh, B. T. Chew, and N. N. N. Ghazali, "A review of human thermal comfort model in predicting human–environment interaction in non-uniform environmental conditions," *J. Therm. Anal. Calorim.*, vol. 147, pp. 14739–14763, 2022, doi: 10.1007/s10973-022-11585-0.

[11] A. Hedge, P. MacNaughton, M. Woo, R. Guglielmetti, and B. Tinianov, "Airport passenger experiences in concourses with either electrochromic or low-e glass windows," *Int. J. Aviat. Manag.*, vol. 5, no. 1, 2021.

[12] H. Huo *et al.*, "Simulation of the Urban Space Thermal Environment Based on Computational Fluid Dynamics: A Comprehensive Review," *Sensors*, vol. 21, no. 6898, pp. 1–27, 2021.

[13] M. H. Al-zuriqat and B. Obeidat, "Computational fluid dynamics – based assessment of thermal comfort parameters in residential buildings in Amman: Implications for indoor environmental quality," *J. Ecol. Eng.*, vol. 26, no. 5, pp. 383–400, 2025.

[14] A. L. Slimani, S. Mazouz, and S. Nekhila, "Computational Fluid Dynamics-Based Quantitative Assessment and Performance Optimization of Thermal Comfort in Hyper-Arid Climate Office Buildings," *Sustainability*, vol. 17, no. 10229, pp. 1–43, 2025.

[15] R. Widiastuti, J. Zaini, M. A. Wibowo, and W. Caesarendra, "Indoor Thermal Performance Analysis of Vegetated Wall based on CFD Simulation," *CFD Lett.*, vol. 12, no. 5, pp. 82–90, 2020.

[16] Z. R. B. Sahari, A. S. A. Nohe, and M. S. Bin Othman, "Thermal comfort in indoor and outdoor spaces: a methodology," *Int. Multidiscip. Acad. Conf.*, 2025.

[17] P. Stiborova, A. Badurova, O. Sikula, and I. Skotnicova, "EVALUATION OF THERMAL COMFORT USING DYNAMIC SIMULATION: A CASE STUDY OF A KINDERGARTEN CLASSROOM IN THE CZECH REPUBLIC," *Slovak J. Civ. Eng.*, vol. 33, no. 3, pp. 20–28, 2025, doi: 10.2478/sjce-2025-0016.

[18] D. Lai *et al.*, "A Comprehensive Review of Thermal Comfort Studies in Urban Open Spaces," *Sci. Total Environ.*, 2020.

[19] L. Wang, M. Ismail, and H. S. Basher, "Energy Efficiency and Comfort Performance of Airport Terminal Buildings: A Systematic Review," *Sci. Technol.*, vol. 33, no. 5, pp. 2357–2394, 2025.

[20] A. Raczkowski, Z. Suchorab, and P. Brzyski, "Computational fluid dynamics simulation of thermal comfort in naturally ventilated room," *MATEC Web Conf.*, vol. 252, no. 04007, pp. 1–5, 2019.

[21] M. M. Othayq, "CFD Investigation on the Thermal

Comfort for an Office Room," *Buildings*, vol. 15, no. 2802, pp. 1–26, 2025.

[22] J. Liu, S. Zhu, M. K. Kim, and J. Srebric, "A Review of CFD Analysis Methods for Personalized Ventilation (PV) in Indoor Built Environments," *Sustainability*, vol. 11, no. 4166, pp. 5–7, 2019.

[23] J. M. Zambrano and L. Baldini, "Integrating CFD and thermoregulation models: A novel framework for thermal comfort analysis of non-uniform indoor environments," *Energy Build.*, vol. 335, 2025.

[24] S. Hossain, A. Abduhu, and S. E. Shad, "An Investigation of Thermal Comfort by Autodesk CFD Simulation at Indoor Living Space in Urban Residential Building in Monsoon Climate," *Int. J. Adv. Res. Publ.*, vol. 3, no. 6, 2019.

[25] A. M. Hanafi, T. A. Abdo, N. A. Abbass, Y. M. Diab, M. G. Abdelfatah, and M. A. Ibrahim, "Optimizing Thermal Comfort and Air Quality in University Classrooms: A CFD- Based Comparative Analysis of HVAC Configurations," *Int. J. Eng. Appl. Sci.*, vol. 2, no. 1, pp. 17–31, 2025.

[26] C. Buratti, D. Palladino, and E. Moretti, "Prediction Of Indoor Conditions And Thermal Comfort Using CFD Simulations: A Study Based On Experimental Data," *Energy Procedia*, vol. 126, no. 201709, pp. 115–122, 2017, doi: 10.1016/j.egypro.2017.08.130.

[27] Y. Xia, T. Xu, C. Shi, L. Tian, T. Zhang, and H. Fukuda, "Research on indoor thermal comfort of traditional dwellings in Northeast Sichuan based on the thermal comfort evaluation model and EnergyPlus," *Energy Reports*, vol. 12, pp. 5234–5248, 2024, doi: 10.1016/j.egyr.2024.11.012.

[28] L. Yang, Z. Chen, and M. Zhen, "Effects of thermal-acoustic interaction on airport terminal's indoor thermal comfort: A case study in cold region of China," *J. Build. Eng.*, vol. 86, p. 108834, 2024, doi: 10.1016/j.jobe.2024.108834.

[29] Azmatullah, B. Suresh, and S. Singh, "Examine Thermal Comfort Inside The Indoor Swimming Pool Through Various Configuration of Inlet and Outlet Vents," *Int. J. Innov. Sci. Eng. Manag.*, vol. 4, no. 1, pp. 46–55, 2025, doi: 10.69968/ijisem.2025v4i146-55.

[30] Y. Kang, H. Yuk, H. H. Jo, and S. Kim, "Indoor thermal environment assessment of a historic building for a thermal and energy retrofit scenario using a CFD model," *Case Stud. Therm. Eng.*, vol. 63, 2024, doi: 10.1016/j.csite.2024.105330.

[31] A. Chourey, P. K. Verma, and P. Shrivastava, "Thermal Comfort Analysis in Dormitory Room by Combined MVHR-Fan Coil," *Int. J. Innov. Sci. Eng. Manag.*, vol. 4, no. 3, pp. 351–363, 2025, doi: 10.69968/ijisem.2025v4i3351-363.

[32] S. Ur, R. Chaudhary, H. Medha, A. Haque, and A. Mahmood, "A Comprehensive Literature Study on Thermal Comfort," *Int. J. Res. Publ. Rev.*, vol. 5, no. 11, pp. 4624–4629, 2024.

[33] M. K. Akyüz, E. Açıkkalp, and Ö. Altunta¸, "Thermal Performance, Indoor Air Quality, and Carbon Footprint Assessment in Airport Terminal Buildings," *Buildings*, vol. 14, no. 3957, 2024.

[34] K. Ratajczak, Ł. Amanowicz, K. Pałaszynska, F. Pawlak, and J. Sinacka, "Recent Achievements in Research on Thermal Comfort and Ventilation in the Aspect of Providing People with Appropriate Conditions in Different Types of Buildings—Semi-Systematic Review," *Energies*, vol. 16, no. 6254, 2023.

[35] T. S. Rajput and A. Thomas, "Computational Fluid Dynamics (CFD) based spatial mapping of indoor air quality and thermal comfort in the indoor environment," *Int. Build. Perform. Simul. Assoc.*, 2023.

.