Coupling Computational Fluid Dynamics and EnergyPlus to Optimize Energy Consumption and Comfort in Air Column Ventilation at a Tall High-Speed Rail Station

Haitao Wang *, Ning Lu, Fanghao Wu and Jianfeng Zhai

School of Environment and Energy Engineering, Anhui Jianzhu University, Hefei 230601, China; ln@stu.ahjzu.edu.cn (N.L.); wfh@stu.ahjzu.edu.cn (F.W.); zjf@stu.ahjzu.edu.cn (J.Z.)

* Correspondence: wht@ahjzu.edu.cn

Abstract: With the rapid development of railways, the air distribution and thermal comfort within waiting halls of high-speed railway stations receive significant attention. In this research, the Energy-Plus and CFD simulation coupling method was employed to investigate three ventilation schemes (column attached ventilation (CAV), side jet ventilation (SJV), column attached with side jet ventilation (CASJV)) for the waiting hall of a high-speed railway station in Guangzhou. The research focused on analyzing the airflow characteristics, thermal comfort, and cooling energy consumption associated with each ventilation method. The results show that thermal stratification phenomena are obvious in summer waiting halls. Most of the predicted mean vote (PMV) values in the research are from −0.5 to 0.5, indicating a comfortable thermal environment. In certain areas of both the CAV and SJV, the LPD1 > 40%, which may lead to a strong sensation of a cold draft for passengers. Compared with the SJV, the CAV and CASJV save 11.89% and 9.25% in cooling energy consumption, respectively. Therefore, the CASJV is more suitable for applications in high-speed railway station waiting halls. The results of this study aim to support the application of this combination of attached ventilation and an “air column” air supply in high-speed railway stations.

Keywords: CFD simulation; EnergyPlus; airflow organization; thermal comfort; cooling energy consumption

1. Introduction

In most countries, building energy consumption accounts for about 40% of total energy use [1]. In building energy consumption, buildings with large spaces tend to have a relatively high proportion of energy consumption. Large-space buildings are characterized by large proportions of building area, high permeability requirements, dense personnel flow, and complex functions, which may lead to strong stacking effects and thermal stratification [2–4]. Especially in summer, high levels of solar radiation and high-temperature walls can lead to high indoor temperatures, resulting in huge energy consumption [5,6]. Simultaneously, the indoor thermal environment of large-space layered air conditioners is varied, which may lead to problems such as a high heat dissatisfaction rate and excessive local wind speed [7]. Therefore, how to improve the indoor thermal environment comfort and reduce the energy consumption of large-scale-space buildings has become one of the focuses in the field of building energy conservation in recent years [8–12]. In traditional ventilation systems for large-scale buildings, hybrid ventilation patterns are frequently employed [13]. However, the hybrid ventilation mode has some limitations in large-space buildings, such as low ventilation efficiency and uneven air distribution [14,15]. To improve internal ventilation, many large transportation buildings have adopted innovative ventilation systems, including “air column” [16,17] technology. This technology uses artificial air columns as the end device of air supply in the building, directs air to the indoor space at a specific angle and direction, realizes air mixing and circulation, and improves indoor...
air quality and ventilation. In addition, the column attached ventilation (CAV) [18,19] is an attached air supply mode suitable for large-space-building air conditioning systems, proposed by Yin et al. This mode combines the characteristics of hybrid and displacement ventilation. The CAV has a wide range of application potential in large buildings and other open spaces [20,21]. It not only improves indoor air quality, but also provides a more uniform temperature distribution and a comfortable indoor environment [22–24]. High-speed rail station spaces usually have a large span, and traditional wall adhesion ventilation has a limited effect in this case. To solve this problem, air columns can be used as a solution. However, relatively few studies have been conducted on combining the CAV with air columns.

Building energy modeling (BEM) and CFD simulations have been extensively employed in numerical simulations of contemporary built environments [25,26]. However, traditional energy consumption simulation software cannot accurately consider the thermal stratification of indoor air, resulting in errors in the simulation results [27]. To address this issue, the researchers attempted to integrate BEM with CFD. This approach aimed to facilitate the exchange of essential information between the two systems, enabling more accurate results compared to using them separately [28,29]. Zhai et al. explored some effective methods for integrating BEM and CFD, such as static coupling and dynamic coupling strategies, and explain the potential of combining the two programs [30]. Zhang et al. used the coupling of the EnergyPlus model and the CFD model to optimize the thermal simulation of large buildings [31]. They found significant differences in airflow simulations between the nodal model and the coupled model. Wang et al. tried to study the performance of natural ventilation systems with EnergyPlus and CFD [32], and found that more accurate predictions can be given based on coupling strategies. Zhang et al. used the coupled simulation method to construct a framework to enhance the prediction of thermal convection on the exterior surface of buildings and optimize the outdoor convective heat transfer models in different urban communities [33]. In addition, Shan et al. combined CFD and building energy simulations to divide large spaces into multiple subregions and calculate the optimal thermostat set point for each subregion [34], showing that the coupling method can effectively provide a thermally comfortable environment in large open-plan offices.

In this study, the thermal environment and energy consumption of the waiting hall of a high-speed railway station in summer were analyzed by combining EnergyPlus and CFD simulation, and the optimal ventilation mode combined with attached ventilation and an air column was explored. Three ventilation schemes were compared in the waiting hall (see Figure 1): the column attached ventilation (CAV), side jet ventilation (SJV), and column attached with side jet ventilation (CASJV). The interior surface temperature of the building was determined by inputting parameters such as the building structure, weather conditions, internal heat disturbances, and the air conditioning system into the EnergyPlus program. On the other hand, the spatial distribution of the thermal environment and airflow in the waiting hall was obtained by CFD simulation. The energy performance of the three ventilation schemes was evaluated by running EnergyPlus, taking into account temperature stratification.
2. Methodology

2.1. Description of EnergyPlus and Fluent

EnergyPlus is a widely used building energy modeling tool whose main function is to assess a building’s energy consumption by simulating its thermal conditions, system performance, and energy efficiency [35]. EnergyPlus uses an integrated synchronization method to enable real-time feedback between building modules, air modules, and building HVAC systems, ensuring accurate simulation results. The software provides key energy and thermal environmental parameters such as surface wall temperature, indoor air temperature, and humidity, which can be used as boundary conditions for CFD simulations. In addition, EnergyPlus’s “Room Air” module can correct the indoor space temperature [36], which is suitable for energy consumption simulation studies of temperature-layered buildings. Therefore, EnergyPlus was chosen in this study to evaluate the energy performance of HVAC systems.

CFD is a simulation technique [37] that integrates principles of fluid mechanics, numerical heat transfer, and computer science. It is utilized to analyze and forecast fluid flow and heat transfer phenomena. As building spaces become more complex, diversified, and large scale, the application of CFD technology in the field of green building design can more accurately predict the built environment and indoor air temperature distribution of the design scheme. In this study, Fluent was used to perform CFD simulations of the indoor environment.

2.2. Coupling Framework of BEM and CFD Programs

The coupling of BEM and CFD simulations can employ two strategies: internal coupling and external coupling [29]. Given the differences in time scale, calculation time, and speed between the two programs, an external coupling strategy was chosen for this study. The external coupling method offers a better balance between calculation time and accuracy during the coupling process. It also provides increased stability and reliability. Hence, the external coupling method was adopted in this study [38–40]. Leveraging the capabilities of EnergyPlus and Fluent software, EnergyPlus provides Fluent with reliable boundary conditions for numerical simulations, and Fluent provides detailed airflow characteristics for EnergyPlus.

Figure 1 illustrates the detailed coupling steps between BEM and CFD. First, the initial conditions, including meteorological parameters, building information, and internal heat disturbances (such as personnel, lighting, equipment, and HVAC systems), are input into EnergyPlus for a non-coupling simulation. This simulation is performed to obtain the internal surface temperature of the building. The obtained internal surface temperature is then utilized as the boundary condition for the CFD simulation to model the indoor environment.
thermal environment. Upon achieving iterative convergence in the CFD simulation, airflow patterns, air temperature, and thermal comfort distributions within the space can be obtained. Following that, the temperature distribution is introduced into the “Room Air” module of EnergyPlus, and an energy consumption simulation is conducted based on the coupling strategy to obtain energy consumption data for the coupled simulation. By employing this coupling process, the thermal environment and energy consumption characteristics of the building can be considered, allowing for a more accurate evaluation and analysis of the three ventilation methods.

**Figure 2.** The coupling framework of BEM and CFD simulation.

### 2.3. Thermal Comfort Indicators

In this study, the overall thermal comfort of the waiting hall was evaluated using the PMV-PPD model, which is widely used in the ISO 7730 [41] standard, to describe and evaluate the thermal environment by the predicted mean vote (PMV) and the predicted percentage of dissatisfied (PPD). The factors affecting the thermal comfort of passengers can be divided into environmental factors (air temperature, relative humidity of air, airflow rate, and average radiation temperature) and human factors (clothing thermal resistance and metabolic rate). Local percentage dissatisfied (LPD) [42,43] is an important indicator for evaluating the comfort of indoor thermal environments. It considers the effects of local thermal and humid environments, including draft (referring to the different degrees of cold sensation in the human body caused by air movement), vertical temperature differences, floor surface temperature, and radiant temperature asymmetry, to measure the degree of dissatisfaction of personnel in a particular area. In this study, the local percentage dissatisfied LPD₁ caused by draft and the local percentage dissatisfied LPD₂ caused by vertical temperature difference were used to evaluate the local thermal comfort of the waiting hall. By considering these factors collectively, we can comprehensively evaluate the thermal comfort of the waiting hall.

\[
LPD₁ = (34 - t_{\text{alt}}) \left(\frac{\bar{v}}{\bar{u}} - 0.05\right)^{0.62} \left(0.37 \cdot \frac{\bar{v}}{\bar{u}} \cdot T_u + 3.14\right)
\]  

(1)

\[
LPD₂ = \frac{100}{1 + \exp(0.856 + 0.576 \cdot \Delta t_{a,p})}
\]

(2)

where \(LPD₁\) is the local percentage dissatisfied caused by draft (%), \(t_{\text{alt}}\) is the local average air temperature (°C), \(\bar{v}/\bar{u}\) is the local average air velocity (m/s), \(T_u\) is the local turbulence intensity (%), \(LPD₂\) is the local percentage dissatisfied caused by vertical temperature difference (%), and \(\Delta t_{a,p}\) is the vertical air temperature difference between the head and ankle (°C).
According to the provisions of GB-T50785-2012 [44], the overall evaluation index and local evaluation index are divided into three levels (Table 1): Grade I represents high thermal comfort, Grade II represents average thermal comfort, and Grade III indicates poor thermal comfort. In this study, the thermal comfort of passengers in the waiting hall was assessed under three different ventilation schemes. By combining this evaluation with the collaborative simulation of EnergyPlus and Fluent, the optimization of thermal comfort could be achieved.

Table 1. Thermal comfort indicators.

<table>
<thead>
<tr>
<th>Grade</th>
<th>Overall Evaluation Indicators</th>
<th>Partial Evaluation Indicators</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>$-0.5 \leq \text{PMV} \leq +0.5$</td>
<td>$\text{PPD} \leq 10%$</td>
</tr>
<tr>
<td>II</td>
<td>$-1 \leq \text{PMV} &lt; 0.5$, $+0.5 &lt; \text{PMV} \leq +1$</td>
<td>$10% &lt; \text{PPD} \leq 25%$</td>
</tr>
<tr>
<td>III</td>
<td>$\text{PMV} &lt; -1$, $\text{PMV} &gt; +1$</td>
<td>$\text{PPD} &gt; 25%$</td>
</tr>
</tbody>
</table>

3. Simulation Case Study

3.1. Description of the Building

In this study, the waiting hall of a high-speed railway station in Guangzhou was selected as the research object, with a waiting hall length of 140 m, width of 58.5 m, height of 21.4 m, internal walls on the east and west sides, glass curtain walls on the north and south sides, and basically symmetrical east-west direction. The size of the air column was about 3.2 m (length) \times 1 m (width) \times 3.5 m (height). The tall structure of the station building leads to a significant cooling load during the summer season, necessitating the implementation of a layered air conditioning system. Due to the huge space of the waiting hall, to save time and improve the calculation accuracy in the subsequent simulation calculation, half of the western part of the waiting hall was selected as the research object in this simulation, and the model is shown in Figure 3.

![Figure 3. Architectural model: (a) surface definition in EnergyPlus modeling; (b) air column distribution in the CFD modeling.](image)

First, EnergyPlus was used for an uncoupled energy consumption simulation, and the EnergyPlus model was set up in detail according to the main initial conditions provided in Table 2. According to GB 50189-2015 [45], in Class A public buildings in hot summer and warm winter areas, the heat transfer coefficient of the envelope structure shall meet the following specified limits: the roof heat transfer coefficient shall not exceed 0.8 K [W/(m²·K)], the heat transfer coefficient of the exterior wall does not exceed 2.5 K [W/(m²·K)]. In this study, the heat transfer coefficient of the roof was 0.654 K [W/(m²·K)], the heat transfer coefficient of the exterior wall was 0.84 K [W/(m²·K)], and the heat transfer coefficient of the exterior window was 1.949 K [W/(m²·K)], all of which met the corresponding limit requirements. The EnergyPlus model was simulated using a full day that represents the hottest month of
summer in Guangzhou. The purpose of the simulation is to predict the temperature of the inside surface of the waiting hall.

Table 2. Settings of EnergyPlus model.

<table>
<thead>
<tr>
<th>Type</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weather data</td>
<td>Typical meteorological year data of Guangzhou</td>
</tr>
<tr>
<td>Architectural models</td>
<td>Figure 3a</td>
</tr>
<tr>
<td>People</td>
<td>960</td>
</tr>
<tr>
<td>Lights</td>
<td>10 W/m²</td>
</tr>
<tr>
<td>Equipment</td>
<td>5 W/m²</td>
</tr>
<tr>
<td>HVAC</td>
<td>CAV, supply air temperature: 21 °C</td>
</tr>
</tbody>
</table>

The EnergyPlus simulation predicted the change in the inside surface temperature in the waiting hall of the high-speed rail station under the scenario of a typical day in summer, as shown in Figure 4. The east wall of the building serves as an insulated boundary. The surface temperature of the interior walls, excluding the roof, remains relatively stable throughout the day, reaching its highest point at 17:00. In contrast, the roof experiences the most significant temperature fluctuations, peaking at 16:00 and reaching a maximum temperature of 40.6 °C. The inner surface temperature of the north and south windows is higher than the wall temperature, reaching its peak at 16:00, due to the radiant heat transfer from the sun. In this study, the temperature of the inner surface under peak load conditions was used as the boundary condition of the CFD simulation, and the waiting hall was simulated.

![Figure 4. Interior surface temperature of waiting hall.](image)

3.2. CFD Modeling

3.2.1. Numerical Process

In the numerical simulation process, the model can be reasonably meshed to effectively capture the details of the flow field [46]. To balance computational accuracy and time cost, the appropriate mesh density needs to be selected based on the complexity of the problem and the limitations of computing resources. In this study, a structured grid was used to refine the grid in the air outlet area of the air column, and the grid dimensions of the side air outlet and the attached air outlet were set to 0.05 × 0.05 m and 0.05 × 0.022 m, respectively. This refinement process is depicted in Figure 5.
Considering the conservation equations of mass, momentum, and energy, and adopting the Discrete Ordinates (DO) model to simulate radiation losses, the Reynolds-Averaged Navier–Stokes (RANS) [47] model is a widely utilized method for simulating turbulence. This model demonstrates favorable computational efficiency and applicability in numerical simulations of airflow within buildings. This study aims to better understand and optimize airflow in buildings by adopting the RANS method to predict the airflow organization characteristics in the air column supply mode. Yin et al. have used experimental data to prove that the SST k-ω model can more effectively calculate the flow field distribution of vertical wall attached jets [48]. After evaluating the merits and drawbacks of various turbulence models and considering the characteristics of the attached ventilation mode, the SST k-ω model is employed in this paper.

### 3.2.2. Boundary Conditions and Settings

Boundary conditions are an important part of CFD simulations that define boundary behavior during fluid flow and heat transfer. Since the coupling method enables data exchange between EnergyPlus and CFD simulations, the inner surface temperature predicted by EnergyPlus was used as the boundary condition of the CFD simulations. The boundary conditions for the CFD simulation are provided in Table 3. The air inlet was specified as a velocity inlet, and the return air was specified as a pressure outlet. The transport equation was discretized using the second-order upwind scheme, and the semi-implicit method for pressure linked equations (SIMPLE) was utilized for the numerical calculation. In terms of convergence criteria, a convergence residual criterion of $10^{-3}$ was used for continuity, momentum, turbulent flow energy, and the turbulent dissipation rate. Additionally, a convergence residual criterion of $10^{-6}$ was used for energy. The Boussinesq hypothesis was employed to consider the impact of variations in air density [49].

**Table 3. Boundary conditions.**

<table>
<thead>
<tr>
<th>Name</th>
<th>The Boundary Type</th>
<th>Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Floor</td>
<td></td>
<td>26.70</td>
</tr>
<tr>
<td>Roof</td>
<td></td>
<td>40.60</td>
</tr>
<tr>
<td>North wall</td>
<td></td>
<td>31.04</td>
</tr>
<tr>
<td>South wall</td>
<td></td>
<td>31.63</td>
</tr>
<tr>
<td>West wall</td>
<td></td>
<td>31.47</td>
</tr>
<tr>
<td>East wall</td>
<td></td>
<td></td>
</tr>
<tr>
<td>North window</td>
<td>Temperature boundary</td>
<td></td>
</tr>
<tr>
<td>South window</td>
<td>Adiabatic</td>
<td>35.23</td>
</tr>
<tr>
<td></td>
<td></td>
<td>35.17</td>
</tr>
</tbody>
</table>
3.2.3. Validation

(1) Grid-independence validation

To mitigate the influence of mesh resolution on the accuracy of simulation results, three different mesh resolutions, namely 1.38 million, 1.53 million, and 1.81 million, were considered. Grid independence was assessed by comparing the velocity distribution of the three grid schemes at different heights, as shown in Figure 6. There is a notable deviation between the results obtained with the 1.53 million and 1.38 million grid schemes. On the other hand, the maximum speed deviation between the simulations conducted with the 1.53 million and 1.81 million grid schemes is 3.7%. Considering the desired accuracy of the numerical results and the computational speed, this study utilized a grid resolution of 1.53 million for the simulations.

![Figure 6. Validation of mesh independence.](image)

(2) Numerical model validation

To evaluate the accuracy and applicability of the SST K-ω model, we compare the nondimensional velocity in the column attached region of the CASJV scheme with the experimental data and calculation formulas [18] in the literature. Figure 7 illustrates the nondimensional velocity attenuation in the attached region. The nondimensional velocity distribution exhibits a good agreement with the experimental results available in the literature, demonstrating a consistent downward trend. However, the simulated data are slightly larger than the experimental data. This is because the column surface generates different degrees of frictional resistance during the experiment, which accelerates the attenuation of the wind speed. Upon comparing the simulation results with the experimental data, it has been found that the average deviation between the simulation and experiment using the SST K-ω turbulence model is 4.85%. The model’s ability to accurately predict the thermal environment of the waiting hall is verified.
To compare the disparities in airflow organization distribution and cooling energy consumption among the CAV, SJV, and CASJV, we made necessary adjustments to the wind speed of each ventilation method. These adjustments were made to ensure that the chosen operating conditions in the activity area exhibit the same temperature distribution characteristics. The specific methods employed for these adjustments are shown in Table 4. In this way, the influence of temperature differences between different schemes on the energy consumption comparison results is eliminated. Consequently, it enables a more accurate assessment of the disparities in cooling energy consumption under different ventilation methods. Scheme 1, scheme 5, and scheme 10 were selected based on a temperature at Y = 1.5 m above the ground and the average temperature of the activity area. The temperature distribution of the three schemes was input into EnergyPlus for energy consumption simulation. This method allows for the evaluation of the effect of different ventilation methods on cooling energy consumption, thereby providing valuable insights for optimizing the ventilation system.

Table 4. Simulation cases of different ventilation schemes.

<table>
<thead>
<tr>
<th>Scheme</th>
<th>Type</th>
<th>Supply Air Temperature (°C)</th>
<th>Velocity of the Attached Jet (m/s)</th>
<th>Velocity of the Side Jet (m/s)</th>
<th>Temperature at 1.5 m (°C)</th>
<th>Average Temperature in the Activity area (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CASJ</td>
<td>21</td>
<td>2</td>
<td>2.6</td>
<td>26.665</td>
<td>26.62</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>21</td>
<td>2</td>
<td>5</td>
<td>27.233</td>
<td>27.31</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>21</td>
<td>2</td>
<td>5.2</td>
<td>27.016</td>
<td>27.087</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>21</td>
<td>2</td>
<td>5.4</td>
<td>26.797</td>
<td>26.858</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>21</td>
<td>2</td>
<td>5.6</td>
<td>26.564</td>
<td>26.65</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>21</td>
<td>2</td>
<td>5.8</td>
<td>26.345</td>
<td>26.457</td>
</tr>
<tr>
<td>7</td>
<td></td>
<td>21</td>
<td>2</td>
<td>6</td>
<td>26.121</td>
<td>26.258</td>
</tr>
<tr>
<td>8</td>
<td>CAV</td>
<td>21</td>
<td>3.6</td>
<td></td>
<td>28.205</td>
<td>27.541</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>21</td>
<td>3.8</td>
<td></td>
<td>27.665</td>
<td>27.001</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>21</td>
<td>4</td>
<td></td>
<td>27.125</td>
<td>26.581</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>21</td>
<td>4.2</td>
<td></td>
<td>26.594</td>
<td>26.036</td>
</tr>
<tr>
<td>12</td>
<td></td>
<td>21</td>
<td>4.4</td>
<td></td>
<td>26.063</td>
<td>25.506</td>
</tr>
<tr>
<td>13</td>
<td></td>
<td>21</td>
<td>4.6</td>
<td></td>
<td>25.532</td>
<td>24.976</td>
</tr>
</tbody>
</table>

Figure 7. Nondimensional velocity comparison between the numerical results for the measurement data (where \( u_m(y^*) \) is the local maximum velocity at a distance of \( y^* \) from the slot outlet, \( u_0 \) is the slot velocity, and \( b \) is the width of the slot) [18].
3.3. Zone Division in Large Space

The thermal stratification of large spaces is caused by heat exchange with the walls, ceilings, and lighting fixtures within the space [11]. In Figure 8, the principle of heat transfer between the building surface and the space is depicted, highlighting three types of heat transfer: conduction from the outer surface to the inner surface $q''_{cond}$, convective heat transfer from the inner surface to the indoor air $q''_{conv}$, and radiant heat transfer from the inner surface to the surrounding wall $q''_{rad}$. To accurately describe the thermodynamic characteristics of the waiting hall in this station, a detailed thermodynamic zoning based on the longitudinal height trend observed in the CFD simulation results is necessary.

$$ q''_{cond} = q''_{conv} + q''_{rad} $$

where $k$ is the thermal conductivity of the ceiling material (W/m·K), $L$ is the ceiling thickness (m), $T_{ic}$ is the exterior surface temperature of the ceiling (K), $T_{ic}$ is the interior surface temperature of the ceiling (K), $h_j$ is the interior convective heat transfer coefficient (W/m²·K), $T_{a}$ is the ambient air temperature (K), $\sigma$ is the Stefan–Boltzmann constant (W/m²·K⁴), $\varepsilon_{ic}$ is the emissivity of the interior surface of the ceiling, $T_{surf(j)}^4$ is the temperature of each of the other surfaces (K), and $F_{ic\rightarrow j}$ is the view factor of the ceiling to other surfaces.

Figure 8. Heat transfer mechanism of the building surface.

4. Results and Discussion

4.1. Air Distribution

By organizing and analyzing the data obtained from the CFD simulations, we can acquire the average air temperatures at different heights, as shown in Figure 9b. With a focus on precise control over the activity area, we selected a greater number of sampling points within the range of 0–2 m, as shown in Figure 9b.

Figure 9a illustrates the average temperature variations across different height sections of the waiting hall under three ventilation schemes. With the increase in height, obvious thermal stratification interfaces appear at 4 m and 18 m. The indoor air temperature rises rapidly at 0–4 m, the temperature gradient is small in the range of 4 m–18 m, and the air temperature rises sharply with the increase in height at 18 m–21.4 m. Based on Figure 9a, the space is divided into three areas from bottom to top in the EnergyPlus program: air conditioning area (0–4 m), middle area (4–18 m), and upper area (18–21.4 m).

Figure 9b illustrates the average temperature variations across different height sections of the activity area (0–2 m) in the waiting hall under three ventilation schemes. The CASJV and CAV prioritize the delivery of fresh cold air to the area of activity due to the adhesion effect, and the average temperature of the cross-section increases as the height increases. Due to the use of only the attached air supply, the CAV results in a significant temperature difference between the head and feet, which could potentially cause discomfort for passengers. The temperature in the activity area of the SJV does not change much,
and the temperature is 26.5–27.5 °C. This is because the nozzle height is 3.5 m, and its high-speed jet has less effect on the activity area.

![Temperature distribution](image)

**Figure 9.** Temperature distribution: (a) the average temperature variations across different height sections; (b) the average temperature variations across different height sections of the activity area.

### 4.2. Thermal Comfort of Personnel in the Activity Area

#### (1) Overall evaluation

Figure 10 illustrates the PMV distribution at Y = 1.5 m for three different air supply schemes. The PMV values between −0.5 and 0.5 in most areas under the three ventilation schemes indicate good thermal comfort in the activity area of the waiting hall. Both the CASJV and CAV utilize attached ventilation, where the airflow is primarily directed towards the activity area through the adhesion effect, delivering fresh cold air. This creates an “air lake” near the ground, ensuring a comfortable environment within the activity area between ventilation columns. However, the air lake area between the air columns will feel slightly cool due to the intersection of air currents, and the PMV value is between −0.5 and −1. In the CAV, the PMV distribution is relatively uniform at Y = 1.5 m, but due to the intersection of airflows, the PMV value exceeds 0.5 in some regions. On the other hand, the CASJV, which employs side air supply, effectively addresses the load in areas distant from the ventilation column, resulting in a PMV value below 0.5 at plane Y = 1.5 m. However, the SJV exhibits the most uneven distribution of PMV values at Y = 1.5 m compared to the other two schemes. This uneven distribution may arise from the use of a single side air supply method, which can cause variations in temperature across different areas near and far from the air column.

#### (2) Local evaluation

The local thermal comfort of the waiting hall was assessed by specifically considering the area near the ventilation column and the air lake area between the air columns. The findings of this evaluation are detailed in Table 5. The temperatures at Y = 0.1 m and Y = 1.7 m were chosen as the ankle and head temperatures, respectively. The results show that there are head and ankle temperature differences in the three schemes: except for the CAV near the air column area of 3.233 °C, the rest of the areas are lower than 3 °C, and LPD2 < 10%, which can provide passengers with a more comfortable riding environment. In the air lake area between the air columns, both the CAV and the SJV give passengers a more pronounced feeling of cold air blowing at the ankle level. In the area near the foot of the ventilation column, the LPD1 of the CAV is much greater than 40%. This is due to the adhesion effect, where cold air is preferentially sent to the activity area, especially
near the bottom of the ventilation column, where the wind speed is large, which may give passengers a strong sensation of a cold draft.

![Figure 10. Y = 1.5 m PMV distribution cloud: (a) CASJV; (b) SJV; (c) CAV.](image)

**Table 5.** Evaluation of local thermal comfort.

<table>
<thead>
<tr>
<th>Area</th>
<th>Type</th>
<th>CASJV</th>
<th>SJV</th>
<th>CAV</th>
<th>SJV</th>
<th>CASJV</th>
<th>SJV</th>
<th>CAV</th>
<th>SJV</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Head</td>
<td>Ankle</td>
<td>Head</td>
<td>Ankle</td>
<td>Head</td>
<td>Ankle</td>
<td>Head</td>
<td>Ankle</td>
</tr>
<tr>
<td>Area near the air column</td>
<td>Local air temperature (°C)</td>
<td>26.24</td>
<td>24.062</td>
<td>27.17</td>
<td>23.937</td>
<td>26.426</td>
<td>26.83</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Local average airflow rate (°C)</td>
<td>0.084</td>
<td>0.74</td>
<td>0.214</td>
<td>1.523</td>
<td>0.146</td>
<td>0.113</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Local turbulence intensity</td>
<td>4.33</td>
<td>11.08</td>
<td>7.134</td>
<td>21.43</td>
<td>6.723</td>
<td>2.77</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The vertical air temperature difference between the head and ankle (°C)</td>
<td>2.178</td>
<td>3.233</td>
<td>−0.404</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LPD2</td>
<td>1.993</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Air lake area</td>
<td>Local air temperature (°C)</td>
<td>25.328</td>
<td>25.236</td>
<td>26.583</td>
<td>25.406</td>
<td>25.787</td>
<td>25.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Local average airflow rate (°C)</td>
<td>0.283</td>
<td>0.383</td>
<td>0.194</td>
<td>0.725</td>
<td>0.44</td>
<td>0.628</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Local turbulence intensity</td>
<td>10.704</td>
<td>4.956</td>
<td>7.38</td>
<td>11.98</td>
<td>18.79</td>
<td>16.78</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The vertical air temperature difference between the head and ankle (°C)</td>
<td>0.092</td>
<td>1.177</td>
<td></td>
<td>0.287</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LPD1</td>
<td>14.975</td>
<td>17.029</td>
<td>8.185</td>
<td>42.794</td>
<td>28.397</td>
<td>42.592</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LPD2</td>
<td>0.34</td>
<td>0.856</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
4.3. Cooling Energy

The temperature stratification data obtained from Fluent were integrated into the EnergyPlus model to calculate the cooling energy consumption for the three ventilation methods. Figure 11 depicts the hourly cooling energy consumption of the three ventilation methods on a typical day during the hottest month of summer. The three ventilation schemes exhibit a similar growth trend in cooling energy consumption throughout the day, with energy consumption increasing with rising temperatures. However, during the time periods of 0:00 to 10:00 and 20:00 to 24:00, the SJV demonstrates significantly higher cooling energy consumption compared to the other two ventilation methods. This is because in the activity area zone, SJV exhibits similar temperature distributions as the CAV and CASJV. However, in the non-air-conditioned areas, the SJV experiences lower temperatures, leading to higher cooling demands and subsequently increased cooling energy consumption. Specifically, the CAV had the lowest cooling energy consumption throughout the entire day, resulting in savings of 2.91% and 11.89% compared to the CASJV and SJV, respectively. Compared to the SJV, the CASJV saves 9.25% in energy consumption throughout the entire day.

![Figure 11. Hourly cooling energy consumption and outdoor temperature.](image)

5. Conclusions

To investigate the application of the air supply method combined with attached ventilation and an “air column” in high-speed rail stations, a large-space waiting hall model was established by coupling EnergyPlus and a CFD simulation, and the airflow characteristics, thermal comfort, and energy consumption under different ventilation schemes (CASJV, SJV, and CAV) were studied. The main findings are summarized as follows:

1. The summer waiting hall exhibits notable thermal stratification. In comparison to the CAV and SJV, the CASJV proves to be effective in actively circulating and blending air within the occupied area. As a result, the CASJV creates a more comfortable environment for passengers.

2. Compared to the SJV, the PMV distribution of the CAV and CASJV is more uniform, and the PMV of the CASJV is smaller. This indicates that the attached ventilation mode can effectively improve the overall thermal comfort of the waiting hall of the high-speed rail station.
(3) All three ventilation schemes resulted in an LPD\(_2\) value of less than 10%, which indicates a comfortable vertical temperature distribution in the activity area. Additionally, compared with the CAV and SJV, the CASJV can effectively reduce the local cold wind feeling.

(4) Under the same temperature distribution characteristics, the CAV and CASJV reduce the cooling energy consumption by 11.89% and 9.25% compared with the SJV. These findings indicate that the CAV and CASJV exhibit improved energy-saving performance in the waiting hall.

**Author Contributions:** Conceptualization, H.W., N.L., F.W. and J.Z.; methodology, H.W. and N.L.; writing—original draft, N.L.; writing—review and editing, H.W.; visualization, F.W.; supervision, J.Z. All authors have read and agreed to the published version of the manuscript.

**Funding:** The authors gratefully acknowledge the financial support for this work provided by the Anhui Provincial Key Research and Development Planning Foundation (202004a07020019, 202004a07020049).

**Institutional Review Board Statement:** Not applicable.

**Informed Consent Statement:** Informed consent was obtained from all subjects involved in the study.

**Data Availability Statement:** The data presented in this study are available on request from the corresponding author.

**Conflicts of Interest:** The authors declare that there are no conflicts of interest. The authors state that they have no known competing financial interests or personal relationships that may influence the work reported in this paper.

**References**


18. Yin, H.G.; Li, A.G.; Liu, Z.Y.; Sun, Y.; Chen, T. Experimental study on airflow characteristics of a square column attached ventilation mode. Build. Environ. 2016, 109, 112–120. [CrossRef]


21. Li, J.; Li, A.; Zhang, C.; Yang, C.; Che, J.; Yin, Y.; Ma, Y.; Hou, Y. Comparative analysis and parametric study of an innovative artificial air column ventilation mode in a high-speed railway station. J. Build. Eng. 2023, 63, 105578. [CrossRef]


27. Pan, Y.Q.; Li, Y.M.; Huang, Z.Z.; Wu, G. Study on simulation methods of atrium building cooling load in hot and humid regions. Energy Build. 2010, 42, 1654–1660. [CrossRef]


34. Shan, X.F.; Luo, N.; Sun, K.Y.; Hong, T.; Lee, Y.K.; Lu, W. Coupling CFD and building energy modelling to optimize the operation of a large open office space for occupant comfort. Sustain. Cities Soc. 2020, 60, 102257. [CrossRef]


41. Olesen, B.W.; Parsons, K.C. Introduction to thermal comfort standards and to the proposed new version of EN ISO 7730. Energy Build. 2002, 34, 537–548. [CrossRef]


48. Yin, H.G.; Li, A.G. Study of attached air curtain ventilation within a full-scale enclosure: Comparison of four turbulence models. *Indoor Built Environ.* 2016, 25, 962–975. [CrossRef]

49. Liu, C.P.; Li, A.G.; Yang, C.P.; Zhang, W. Simulating air distribution and occupants’ thermal comfort of three ventilation schemes for subway platform. *Build. Environ.* 2017, 125, 15–25. [CrossRef]

**Disclaimer/Publisher’s Note:** The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.