Numerical Study of Thermal Comfort Evaluation in Naturally Ventilated Classrooms

Bowo Yuli Prasetyo*, aRizki Muliawan, aAndini Luthfiyyah Afifah, bRachmad Imbang Tritjahjono, cFujen Wang

aDepartment of Refrigeration and Air Conditioning Engineering, Politeknik Negeri Bandung, Bandung 40559, Indonesia
bDepartment of Mechanical Engineering, Politeknik Negeri Bandung, Bandung 40559, Indonesia
cDepartment of Refrigeration, Air Conditioning, and Energy Engineering, National Chin-Yi University of Technology, Taichung 411, Taiwan

Received July 15, 2023; Revised August 30, 2023; Accepted September 12, 2023; Published October 1, 2023

ABSTRACT
The classroom is one of the important facilities in teaching and learning activities. Just like any other room, ensuring thermal comfort becomes a vital aspect for its occupants. This study aims to evaluate environmental conditions and the thermal comfort experienced by classroom occupants through numerical methods. The evaluation involves the integration of the Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD) equations into Computational Fluid Dynamics (CFD) simulations using a user-defined function. Furthermore, extensive field measurements and questionnaire surveys are also employed to support the simulation process. The results indicate that the air temperature tends to be high, reaching 30.91°C. Meanwhile, the air velocity within the room, particularly in the occupied areas, is significantly low, with an average value of 0.04 m/s. According to the ASHRAE Standard 55, the classroom is perceived as uncomfortable, as evidenced by a PMV value of 1.38 and a PPD of 44.94%. Therefore, efforts are required to enhance the thermal comfort in the space.

KEYWORDS
Thermal comfort
Natural ventilation
Predicted Mean Vote (PMV)
Percentage of Dissatisfied (PPD)
Computational Fluid Dynamics (CFD)

INTRODUCTION
The classroom, an essential space in educational institutions such as schools, universities, and training centers, plays a crucial role in the teaching and learning process [1,2]. Various requirements must be fulfilled for a room to function effectively as a classroom, including sufficient space, proper ventilation, adequate lighting, and optimal temperature to ensure comfort [3,4]. Comfort within the classroom setting is essential as it directly impacts the performance and productivity of its occupants [5]. A supportive environment promotes students’ engagement with learning materials, facilitates interactions, and enhances the achievement of learning objectives. Unfortunately, the occurrence of hot and uncomfortable classroom conditions is a common issue that disrupts the concentration and productivity of the occupants daily [6].
This evaluation aims to assess and analyze the thermal comfort experienced by occupants in the room, considering variables such as air temperature, air velocity, and relative humidity, utilizing numerical simulation methods. The findings from this research are expected to serve as a foundation for designing a more effective and optimal thermal management strategy in the Refrigeration and Air Conditioning Engineering Department Building.

**LITERATURE REVIEW**

Thermal comfort refers to the conditions of temperature, humidity, and airflow in the environment that impact an individual’s physical and psychological comfort level [5,6]. Standards for thermal comfort often refer to the ASHRAE 55 standard, which considers various parameters such as air temperature (19.6°C - 27.9 °C), air velocity (0.2 m/s - 0.8 m/s), relative humidity (60%), clothing insulation, and metabolic rate [7]. Two commonly used thermal comfort indices for evaluating comfort are the Predicted Mean Vote (PMV), with a recommended range of -0.5 to +0.5, and the Percentage of People Dissatisfied (PPD), with a threshold of less than 10% [8,9].

According to a study, the condition of temperature and humidity inside the room significantly impact the comfort level of its occupants [10-13]. Evaluating thermal comfort can be achieved through field investigations, including questionnaire surveys, and instrumental measurements of thermal comfort variables. Common variables used as references for assessing thermal comfort include air temperature, air velocity, and relative humidity [13,14].

Numerical simulation is a mathematical computational method utilized to model and analyze complex systems [15,16]. In the simulations, complex systems are transformed into simplified mathematical models with equations and geometry. The objective of CFD is to provide accurate predictions regarding fluid flow, heat transfer, and chemical reactions in intricate systems [17,18]. This approach, known as Computational Fluid Dynamics (CFD), involves implementing the model in a computer program to generate numerical solutions that closely approximate real-world conditions [17].

Numerical simulation offers several advantages, including the flexibility to model diverse system types, the ability to conduct virtual experiments without physical risks, in-depth analysis of numerical results, time and cost savings compared to experimental approaches, predictive and planning capabilities, and better conceptual understanding of complex phenomena [18]. Consequently, numerical simulation has emerged as a powerful method for effectively and efficiently analyzing, understanding, and developing systems.

Buratti et al. conducted a study to evaluate thermal comfort sensation using experimental data and CFD codes [19]. Experimental data is used to validate the CFD model which is then used to predict global and local thermal comfort in the room. Yadeta et al., in their research, mentioned that the results of thermal comfort in the building did not meet the standards set. This was found in a study of buildings in Jimma City that caused long-term discomfort for residents due to inappropriate design and construction. Based on CFD simulations, a residential house should involve adaptive mechanisms such as opening windows, changing fabrics, or insulating internal windows to improve thermal comfort [20]. Hajdukiewicz et al. conducted a study stating that to create effective natural ventilation in buildings, a good understanding of the complex airflow patterns influenced by buoyancy and wind is required [22,23].
RESEARCH METHOD

This research was conducted in the Refrigeration and Air Conditioning Engineering Department Building, located in Bandung, Indonesia, which has a tropical climate, as shown in Figure 1. The building comprises three main floors along with a rooftop. It is oriented eastward at the front and westward at the back. For this study, the focus was on a specific classroom located on the 3rd floor of the building, which is regularly used for teaching and learning purposes. The room accommodates 30 students and one lecturer. This particular classroom was chosen as a sample due to its standardized dimensions and layout, representative of other classrooms in the building.

NACA Airfoils

Measurements were conducted for 1.5 hours, starting from 13:00 to 14:30, while the classroom was in use. The room accommodated 30 students and 1 lecturer during the measurement period. The door remained closed while the windows were open, allowing for natural ventilation. The field measurements were essential for obtaining the necessary parameters to determine indoor thermal comfort, including air temperature, air velocity, and relative humidity. The measurement results include environmental conditions, building and room characteristics, as well as design data used in creating simulation models [21]. The measuring instruments employed in this study are outlined in Table 1. While the visual representation of the measurement points and sensor placement is shown in Figure 2.
Table 1. Specification of measuring device

<table>
<thead>
<tr>
<th>Measuring Device</th>
<th>Type/Model</th>
<th>Range Operation</th>
<th>Resolution</th>
<th>Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature Data Logger (Tc type-K)</td>
<td>Pico Technology, USB TC-08</td>
<td>0 °C-50 °C</td>
<td>-20 °C</td>
<td>±0.2 % ±0.5 °C</td>
</tr>
<tr>
<td>Humidity Meter HT -3006A</td>
<td></td>
<td>10 % to 95 % RH</td>
<td>0.1 %</td>
<td>≥70 % RH - ± (3 % reading + 1 % RH)</td>
</tr>
<tr>
<td>Hot Wire Anemometer AM-4204</td>
<td></td>
<td>0.2 - 20.0 m/s</td>
<td>0.1 m/s</td>
<td>±(5 % + 1 d)</td>
</tr>
<tr>
<td>Thermometer Gun Raytek (MT4)</td>
<td></td>
<td>-18 to 400 °C</td>
<td>0.2°C</td>
<td>-18 °C (0°F) to -1 °C (30°F) ±3°C (±5°F)</td>
</tr>
<tr>
<td>Thermometer Digital APPA50 (50II Series)</td>
<td></td>
<td>-200 °C ~ 1372 °C</td>
<td>0.1 °C</td>
<td>T1-T2 ± (0.3% °F T1-T2 reading+2.0°C),±(0.3% °F T1-T2 reading+4.0°F)</td>
</tr>
</tbody>
</table>

In addition to field measurements, a questionnaire survey was also conducted to gather subjective data, including information on clothing insulation and the activities of the room occupants. These data were crucial for calculating the level of thermal comfort. The survey involved 31 occupants of the room, and the list of questions can be found in Table 2 below.

Table 2. Survey questionnaire

<table>
<thead>
<tr>
<th>Survey Questionnaire</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gender of Respondent</td>
</tr>
<tr>
<td>Respondent’s Seating Position</td>
</tr>
<tr>
<td>Respondent’s Body Mass</td>
</tr>
<tr>
<td>Respondent’s Body Height</td>
</tr>
<tr>
<td>Respondent’s Activity Type</td>
</tr>
<tr>
<td>Clothing used by the Respondent (Top)</td>
</tr>
<tr>
<td>Clothes used by Respondents (Bottoms)</td>
</tr>
<tr>
<td>Footwear</td>
</tr>
</tbody>
</table>
Numerical Method

The CFD simulation was conducted using the steady-state method. ANSYS Fluent software was employed for the CFD simulation, utilizing the k- ω SST turbulence model equation. This specific model was chosen due to its improved accuracy and reliability for cases involving stronger flow conditions compared to the standard model [15]. The k- ω SST model exhibits a similar form to the standard k-w model, with the governing equations depicted in equations (1) and (2) as follows [16,15].

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{2}
\]

The room geometry used for simulation purposes is depicted in Figure 3. The classroom has a total area of 60.72 m². The front wall of the room is identified by the presence of a blackboard. On the left wall of the room, there is another glass structure and window openings that are parallel to the door. The door is oriented towards the north, causing the section of the room mostly occupied by the glass to face the south. The window openings, which are parallel to the door and facing inward into the building, are assumed to function as outlets. Additionally, there are six lamps positioned at the top of the room, which emit heat when turned on. The heat flux value for the front and back walls is assumed to be zero, as the temperature of the adjacent rooms is considered to be the same. The representation of human occupants in the simulation is simplified to reduce the simulation’s load, while still ensuring that the number and positioning of human objects are appropriate.
The specific boundary conditions used in the study are presented in Table 3. The values for human heat flux are determined based on previous scientific journals [18], while the inlet and outlet conditions are obtained from actual data collected during field measurements.

<table>
<thead>
<tr>
<th>Kind of Rooms</th>
<th>Boundary Condition</th>
<th>Type</th>
<th>Value</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classroom</td>
<td>Inlet</td>
<td>Velocity</td>
<td>Velocity: 0.17 m/s</td>
<td>(6.66 x 0.20 x 0.4) m</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Inlet</td>
<td>Temperature: 29.4°C</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Outlet</td>
<td>Velocity</td>
<td>Velocity: 0.20 m/s</td>
<td>(4.44 x 0.20 x 0.85) m</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Outlet</td>
<td>Temperature: 30.22°C</td>
<td></td>
</tr>
<tr>
<td>Human</td>
<td>Wall</td>
<td></td>
<td>18 W/m²</td>
<td></td>
</tr>
</tbody>
</table>

Three mesh variations with different numbers of elements are used to carry out grid independence studies and validate CFD model solutions according to room conditions. The difference in the number of elements between each variation, which is referred to as the refinement ratio, must be greater than 1.30 [18]. The equation for determining the refinement ratio can be seen in equation. Based on these equations, meshing results are obtained, with the variations presented in Table 4.

$$r = \left( \frac{\Delta \text{fine}}{\Delta \text{coarse}} \right)^{\frac{1}{3}}$$

<table>
<thead>
<tr>
<th>Variations</th>
<th>Meshing</th>
<th>Elements</th>
<th>Nodes</th>
<th>r</th>
<th>Skewness</th>
<th>Orthogonal Quality</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Coarse</td>
<td>398735</td>
<td>79447</td>
<td>-</td>
<td>$1.14 \times 10^{-3}$</td>
<td>0.25-0.99</td>
</tr>
<tr>
<td>2</td>
<td>Medium</td>
<td>939688</td>
<td>178600</td>
<td>1.33</td>
<td>$5.17 \times 10^{-4}$</td>
<td>0.28-0.99</td>
</tr>
<tr>
<td>3</td>
<td>Fine</td>
<td>2385272</td>
<td>444835</td>
<td>1.36</td>
<td>$1.02 \times 10^{-3}$</td>
<td>0.24-0.99</td>
</tr>
</tbody>
</table>

The PMV and PPD equations are introduced into the simulation, based on International Standards [10]. The PMV index is included to enable the simulation by entering equations (4) to (8) into the User-Defined Function (UDF).

After the comparison of each alternative, the next step is to do a material rating. The ranking is obtained by calculating the total global priorities [22]. Based on Table 3, the selection of alternative materials for outer support solar water heaters using the Analytic Hierarchy Process (AHP) approach, it is found that AISI 201 stainless steel material is the best material that can be chosen for the manufacture of outer support by having a combination of strength criteria, corrosion resistance, ease the of process, hardness, and material costs that are better than other materials [23].
Table 5 Ranking of Material

<table>
<thead>
<tr>
<th>Material</th>
<th>Total Global Priorities</th>
<th>Ranking</th>
</tr>
</thead>
<tbody>
<tr>
<td>AISI 201</td>
<td>0.2705</td>
<td>1</td>
</tr>
<tr>
<td>AISI 316</td>
<td>0.1716</td>
<td>4</td>
</tr>
<tr>
<td>AISI 304</td>
<td>0.2055</td>
<td>3</td>
</tr>
<tr>
<td>Aluminum 502</td>
<td>0.0917</td>
<td>5</td>
</tr>
<tr>
<td>AISI 1020</td>
<td>0.2605</td>
<td>2</td>
</tr>
</tbody>
</table>

Figure 4 displays the validation results of the simulation involving three different mesh variations, compared to the field measurements. The figure clearly shows that mesh variation 2 produces results that closely align with the measurement data, with average differences of 0.36% for temperature and 3.22% for air velocity. Based on these findings, mesh variation 2 is selected for further evaluation.

Figure 4. Validation of simulations and field measurement: (a) Temperature (b) Air velocity

Figure 5 illustrates the contour of air temperature inside the classroom. Three different perspectives are shown, representing an overall depiction of the room conditions. The simulation results indicate that the temperature within the classroom ranges from 29°C to 32°C, with an average value of 30.91°C. In the top view visualization Figure 5 (a), it can be observed that the temperature near humans tends to be warmer compared to the surrounding areas. This is due to humans being a source of heat within the room. A similar observation can also be made in the side view (Figure 5 (c)), where the temperature near the lamp appears to be higher.

Figure 5. Visualization of temperature contour: (a) Plane 1, (b) Plane 2, (c) Plane 3

Figure 6 depicts the simulation results of air velocity distribution within the classroom. The visualizations reveal an uneven distribution of air velocity. In the top view visualization (Figure 6 (a)), the average air velocity around the occupants ranges from 0 to 0.05 m/s. However, in the
front view visualization (Figure 6 (b)), higher air velocities are observed in the upper part of the room. The presence of objects and occupants within the space limits the airflow, causing most of the air to circulate at the top of the room. These low air velocities adversely affect the thermal comfort of the room occupants [24].

This observation is further supported by the visualization presented in Figure 6. The simulation results indicate that the outdoor air entering the room through the windows tends to flow toward the upper part of the room. A small portion of the air then descends towards the occupants before exiting through the outlets. Meanwhile, the majority of the air moves directly toward the outlets. The inflow of outdoor air is a crucial factor in a room with natural ventilation. This air typically has a lower temperature than the air inside the room, providing a more comfortable environment for the occupants.

Figure 7 depicts the simulation results of air velocity distribution within the classroom. The visualizations reveal an uneven distribution of air velocity. In the top view visualization (Figure 7 (a)), the average air velocity around the occupants ranges from 0 to 0.05 m/s. However, in the front view visualization (Figure 7 (b)), higher air velocities are observed in the upper part of the room. The presence of objects and occupants within the space limits the airflow, causing most of the air to circulate at the top of the room. These low air velocities adversely affect the thermal comfort of the room occupants [24].

This observation is further supported by the visualization presented in Figure 7. The simulation results indicate that the outdoor air entering the room through the windows tends to flow toward the upper part of the room. A small portion of the air then descends towards the occupants before exiting through the outlets. Meanwhile, the majority of the air moves directly toward the outlets. The inflow of outdoor air is a crucial factor in a room with natural ventilation. This air typically has a lower temperature than the air inside the room, providing a more comfortable environment for the occupants.
Figure 7. Visualization of velocity streamline: (a) Isometric, (b) Plane 2, (c) Plane 3

Figure 8 illustrates the contour of PMV and PPD values distributed throughout the classroom. The simulation results indicate that the PMV values range from 0.9 to 2, with the highest values scattered in the occupied areas, while the lowest values are observed near the inlet. These values indicate that the classroom is in a warm condition according to the ASHRAE standard [9]. This is further supported by the PPD contour, which shows an average room value of 44.49%. This value exceeds the ASHRAE standard’s comfort range of 5-10%. These findings can be attributed to the relatively high room temperature and the insufficient air movement in the occupied areas, limiting the thermal comfort of the occupants.

CONCLUSION

In conclusion, this study conducted a comprehensive analysis of thermal comfort in the classroom environment. Numerical simulations provide valuable insights into various aspects, including temperature distribution, air velocity, and comfort parameters such as PMV and PPD. The
simulation results show that the average temperature in the room is 30.91 °C. The uneven air
distribution indicates low air movement over occupied territory, with an average value of 0.04
m/s. These findings indicate that the thermal conditions in the classroom do not meet the comfort
requirements specified by the ASHRAE standard.

The PMV and PPD values further confirm this finding, with the average PMV and PPD values
being 1.39 and 44.49%, respectively. These results highlight the need to improve the thermal
comfort conditions in the classroom. Possible measures include increasing air circulation and
optimizing ventilation systems. Implementing these measures can significantly increase occupant
comfort and promote a conducive learning environment. Further research and interventions are
needed to achieve optimal levels of thermal comfort in educational buildings.

ACKNOWLEDGEMENTS

The authors would like to express their great appreciation for the financial support from
Politeknik Negeri Bandung.

REFERENCES

comfort in Spanish primary classrooms during summer season,” vol. 203, 2021, doi:
Experimental and numerical studies on indoor thermal comfort in fluid flow : A case
10.1016/j.proeng.2017.03.027.
optimization of thermal comfort in building : Central composite design and CFD
Field investigations on thermal comfort in university classrooms in,” Energy Reports,
with a new ventilation system,” Build. Environ., p. 107016, 2020, doi:
conditions in a classroom equipped with radiant cooling systems and subjected to
uniform convective environment,” vol. 35, pp. 1292–1305, 2011, doi: