THERMAL COMFORT ANALYSIS IN AN EDUCATIONAL BUILDING IN A HOT CLIMATE AREA USING CFD

[1]Aruna MOHAN* [2]Velkennedy R [3]Rajkumar DEIVENDIRAN

[1]Assistant Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai-15, India Email : maciv@tce.edu

[2]Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai-15, India Email: rvkciv@tce.edu

[3]Assistant Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai-15, India, Email: rajkumarcivil@tce.edu

* Corresponding author:
Ms.Aruna Mohan, Assistant Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai, Tamilnadu, India-625015. Mail id : maciv@tce.edu

Abstract

This study uses Computational Fluid Dynamics (CFD) simulations to optimize the indoor thermal environment in a computer lab at a civil engineering department in Madurai, Tamil Nadu, India, characterized by a hot semi-arid climate by evaluating different roof and Heating, Ventilation, and Air Conditioning configurations using CFD simulations and to examine the thermal comfort and indoor air quality. The goal of the study is to comprehend how various roof materials and air conditioning affect the temperature, air dispersion, and thermal comfort of the room. Autodesk Revit was used to construct three scenarios, while Autodesk CFD and ANSYS Fluent were used for analysis. The findings of the study provide the use of gypsum-board roofing combined with AC for optimizing thermal performance for the design of educational facilities, since they emphasize how important air conditioning and material choices are to attaining the best possible thermal comfort. Scenario III (AC + Gypsum roof) showed a 24% temperature drop compared to Scenario I (RCC roof), and 15% better thermal distribution than Scenario II.

Keywords: CFD, Thermal Comfort, Educational Buildings, Hot Climate, Indoor Air Quality, Computer Lab

1. Introduction

Student productivity enhancement and their well-being in educational institutions relies on thermal comfort and indoor air quality. It affects cognitive performance negatively and creates discomfort if the airflow is poorly distributed. Factors influencing thermal comfort according to ASHRAE 55 standards[5] are air temperature, mean radiant temperature, humidity, air velocity, clothing metabolic rate and insulation.

The thermal quality of indoor environments plays a fundamental role in supporting student comfort, health, and academic effectiveness, particularly in educational buildings. This is especially true for computer labs, where high occupant density, continuous use of electronic devices, and limited ventilation can exacerbate thermal stress. Environments with poor airflow and excessive heat buildup can lead to discomfort, lower concentration levels, and reduced learning performance.

Maintaining thermal comfort in such settings requires careful control of several interacting factors, including air temperature, humidity, radiation from surfaces, airflow velocity, clothing insulation, and metabolic rate. These elements are identified in international comfort standards, such as ASHRAE 55[5]. In academic settings, users often remain stationary for long durations, further intensifying their sensitivity to minor fluctuations in thermal conditions. Ensuring acceptable thermal conditions is therefore critical not only for occupant comfort but also for protecting sensitive electronic equipment and improving space usability.

To analyze and optimize the indoor environment, Computational Fluid Dynamics (CFD) has become a valuable tool. CFD enables the simulation of air movement, heat transfer, and pollutant dispersion inside enclosed spaces, supporting the development of efficient ventilation strategies and HVAC (Heating, Ventilation and Air Conditioning) designs. Through virtual modeling, researchers can study various material options and system configurations to determine their impact on thermal comfort and air quality. Recent literature emphasizes the positive effect of improved indoor environments on user performance, further highlighting the importance of such studies in academic infrastructure.

This research is centred on a real-world case study: a computer lab situated within the Civil Engineering Department of Thiagarajar College of Engineering in Madurai, Tamil Nadu, India. The city experiences a hot semi-arid climate, posing additional challenges in maintaining indoor comfort. The lab was chosen for its consistent use, equipment density, and relevance as a representative environment for thermal analysis.

The study specifically investigates the influence of ceiling materials on indoor thermal behavior. In this context, the term “roof material” refers to the ceiling's inner surface layer, which significantly affects heat transfer between the roof and the room interior. Three design cases were developed and assessed using Autodesk Revit for modeling and simulated using Autodesk CFD and ANSYS Fluent. The cases include: (1) a reinforced concrete (RCC) slab without air conditioning, (2) a gypsum ceiling with natural ventilation (fans), and (3) a gypsum ceiling combined with active cooling (AC). Simulations were conducted for peak summer (July) and moderate winter (December) conditions[9].

The goal of this paper is to provide detailed insights into how ceiling construction and HVAC strategies influence temperature distribution and air quality in educational spaces. By applying CFD-based simulation, the study delivers a quantitative understanding of airflow patterns and thermal comfort metrics that can inform future classroom and lab designs.

With the advancements in technology, CFD (Computational Fluid Dynamics) allows for the prediction and optimization of air quality criteria and conditions. The analysis of air circulation and temperature distribution in different buildings and types of HVAC systems is simplified through CFD. CFD has been shown to work well in assessing air quality and the conditions related to the comfort of occupants in various indoor settings. The study focuses on a computer laboratory within a civil engineering department. Three scenarios were modelled using Autodesk Revit and analyzed with Autodesk CFD and ANSYS Fluent. The scenarios include variations in roof materials and the presence of air conditioning. The results are intended to inform the design of educational facilities to achieve optimal thermal comfort.

The findings of this study have significant implications for the design and operation of educational facilities. By understanding the impact of roof materials (*refers to the internal ceiling covering which impacts thermal insulation and heat gain*) and air conditioning on thermal comfort and IAQ, designers and engineers can make informed decisions that enhance the indoor environment. This can lead to improved occupant comfort, increased energy efficiency, and reduced operational costs. Additionally, the use of CFD simulations provides a powerful tool for visualizing and optimizing indoor environmental conditions, offering a detailed and accurate assessment that can guide future research and practice.

2. Literature Review

Computational fluid dynamics analysis of a building plan has been investigated by Obula Reddy Kummitha et al. (2021)[3] with predominant wind velocity for different wind directions. The flow properties‟ variation in the computational domain has been modeled by solving the Reynolds-Averaged Navier–Stokes(RANS) equations with the finite volume second-order discretization scheme. The turbulence of airflow distribution in and around the building has been modeled with the Shear Stress Transport turbulence model from the analysis of different turbulence models. Numerical results are analysed by evaluating and comparing the various flow properties at different building plan locations with different wind directions.

The analysis in Ashish Mogra et al. (2018) [3] accounts for the relationship between productivity and thermal comfort and indoor air quality within the context of a classroom. The researchers analyze how the airflow from an air conditioner in the classroom is distributed using CFD simulation. The study proposes a new design of an air-conditioned classroom whereby the air conditioner is mounted diagonally instead of centrally located, and performs simulations to test this hypothesis. The findings illustrate a high degree of concentration index of students and professors on where the air conditioning unit was placed to provide comfortable air velocities and temperatures. Zhou et al. (2024)[12]. The main aim of this research is to develop mathematical models to analyze surface radiation and natural convection heat transfer in a solar cavity receiver.

A comprehensive numerical investigation into the combined effects of natural convection, conduction, and surface radiation within a sealed square cavity containing two heat-generating electronic components modeled as square blocks was studied by Hidki et al (2022)[8]. The study explores how different configurations—placing the blocks at the same height (horizontal position) or at different heights (vertical position)—along with varying parameters such as Rayleigh numbers, thermal conductivity ratios, and wall emissivity influence the flow patterns and temperature distributions inside the cavity. By analyzing streamlines, temperature profiles, and heat transfer rates, the research highlights how the positioning of components and radiative properties significantly impact cooling efficiency. The findings reveal that increasing temperature differences and radiative emissivity can enhance heat dissipation, thereby lowering the maximum temperatures of the electronic blocks. Overall, the study provides valuable insights for optimizing electronic cooling systems by understanding the interplay between natural convection, conduction, and radiation in confined environments containing multiple heat sources.

3. Methodology

The governing equations were discretized using the Navier-Stokes equation. Autodesk Revit was employed to create a 3D model of the existing laboratory based on the collected initial data, including room parameters and dimensions. This 3D model was then exported and imported into Autodesk CFD 2021 software for various analyses. Subsequently, the model was analyzed in ANSYS Fluent to study the thermal flow within the laboratory. The commercial CFD software ANSYS Fluent is a leading tool for fluid flow analysis, capable of simulating heat transfer, mass transfer, turbulence, and radiation effects with high accuracy. It was selected because of its robustness, validated numerical solvers, and ability to handle complex boundary conditions relevant to indoor environments. The case study site that was chosen is the Computer Laboratory of the Educational Institution which is depicted in figure.1. The described configuration creates classroom conditions that allow for examination of different roof types, air conditioners, and their influence on thermal comfort and air quality using Computational Fluid Dynamics (CFD) modeling. The components placed in the room and the dimension details are listed in table 1.

To accurately capture airflow and temperature distribution, the governing fluid dynamics were modeled using the Navier–Stokes equations. These equations describe the conservation of mass, momentum, and energy in fluid flows and are essential for predicting complex interactions between air motion and heat transfer within enclosed spaces. The general form of the incompressible Navier–Stokes equation used in this study is:

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) = -\nabla p + \mu\nabla^2\mathbf{u} + \mathbf{f}$$

where:

- "ρ is the fluid density"
- "$\mathbf{u}$ is the velocity vector"
- "p is the pressure"
- "μ is the dynamic viscosity"
- "$\mathbf{f}$ represents body forces (e.g., gravity)"

This equation was applied during the CFD simulation phases to model the airflow and temperature distribution within the room, with the assumption of incompressible, steady-state flow and buoyancy-driven natural convection.

Table 1 Components in Laboratory

Parameter	**Details**
Room Dimension	40’ x 30’
Room Area	1200 sq.ft
Natural Convection Windows	5 Nos (6’ x 5’)
Window Material	Iron
Doors	2 Nos (10’ x 4’)
Door Material	Aluminium and glass glazing
Forced Convection Fans	7
Pedestal Fans	2
Wall Mounted Fans	4
Heat Source – Computers	43
UPS and Batteries	Yes
Light Sources	10

Figure 1: Study area

3.2 Data Collection

Room temperature measurements were conducted using an Air Quality monitor. Figure 2a is the Air quality monitor used for measuring room temperature data. Eight specific points within the room were selected for data collection. Figure 2b represents the plan of the layout of the room, with marked positions of the fans, and 8 measuring points. A grid was established in the room, and temperature readings were taken at both the boundary and intermediate points. Specifically, temperatures were recorded at the four corners of the room and four intermediate points within the grid. These measurements were taken during July and December to capture seasonal variations. The respective data are given in table 2.

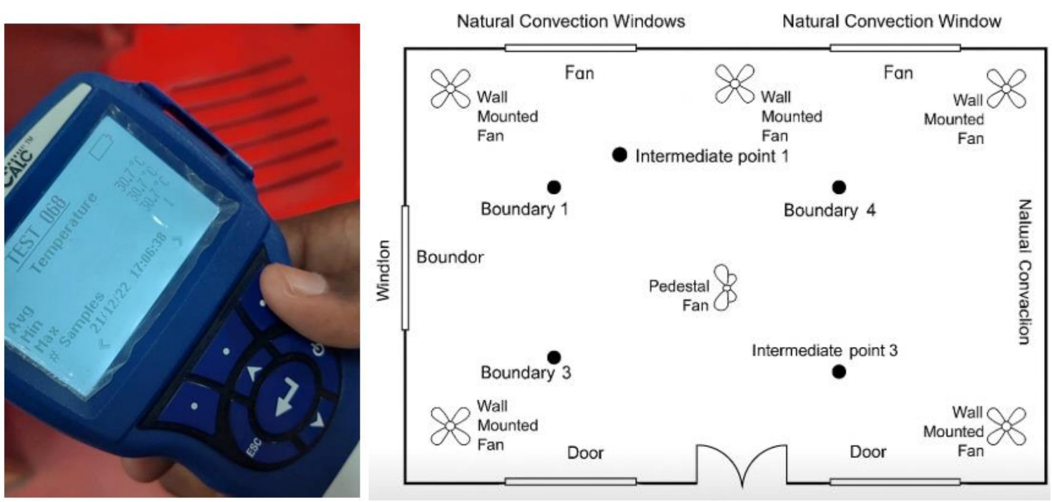

Figure 2a: Air Quality Monitor 2b: Layout of the room

Table 2: Temperature data for the July and December months

	Particulars	July Temperature (°C)	December Temperature (°C)
1	Boundary 1	33.5	31.8
2	Boundary 2	33.3	31.3
3	Boundary 3	33.2	30.9
4	Boundary 4	33.6	31.3
5	Intermediate point 1	33.2	30.7
6	Intermediate point 2	31.2	30.7
7	Intermediate point 3	33.2	30.7
8	Intermediate point 4	33.5	30.8

3.3 Autodesk CFD Modelling

For purposes of the simulation analysis, the Case Study room is imported in Autodesk CFD software. The CFD software takes care of the Materials, Boundary Conditions, and Mesh designing. To assign the materials, the model is brought into Revit 3D and the materials are imported. Boundary conditions also have to be determined for the already set materials. Figure 4 represents the setting of the boundary condition in the software.

Creating a mesh is important because the Navier-Stokes equations can only be solved analytically for very simple flows under stringent conditions. In practical situations, a numerical approach is needed in which the algebraic expression of the system's governing equations is solved through sequential calculation steps instead of analytically. The figure 3 shows the mesh sizing.

The discretization involves a continuous spatial domain which is transformed into a finite collection of control volumes. Each control volume is a reasonably small subsection of the whole domain, the governing equations are applied in an integrated fashion with respect to these volumes. These methods guarantee that fundamental physical properties, like mass, momentum, and energy, in a disaggregated form, remain balanced in each control volume.

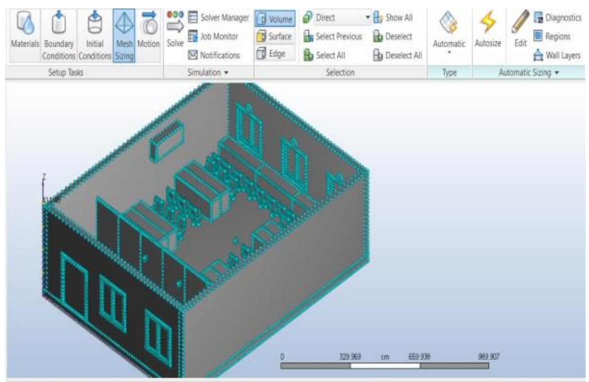

Figure 3: Meshing

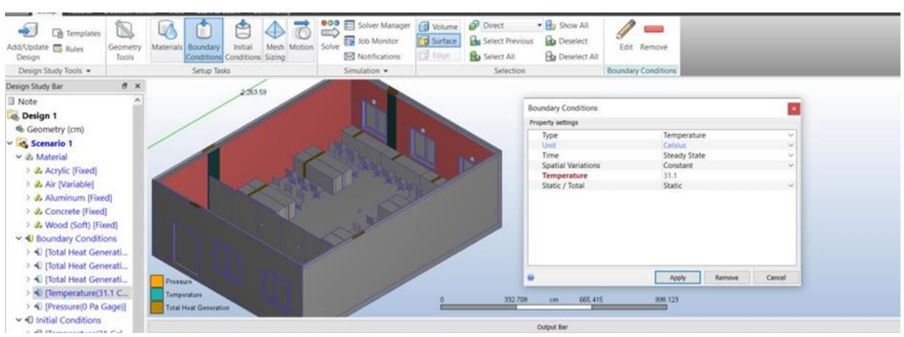

Figure 4: Setting boundary condition

3.4 Scenarios

The Laboratory is designed in Autodesk Revit and analyzed in three scenarios.

3.4.1 Scenario I

In Scenario I, the natural convection in the room is analyzed based on the temperature data collected. The roofing material used in the study is Reinforced Cement Concrete (RCC), a widely used composite material composed of concrete and steel reinforcement bars, offering high structural strength and durability. However, RCC alone exhibits poor thermal insulation, leading to significant heat gain in hot climates. The inputs are fed into Autodesk CFD to run the CFD analysis fluently in

ANSYS. Figure 5 shows scenario I along with cross sectional view of the roof.

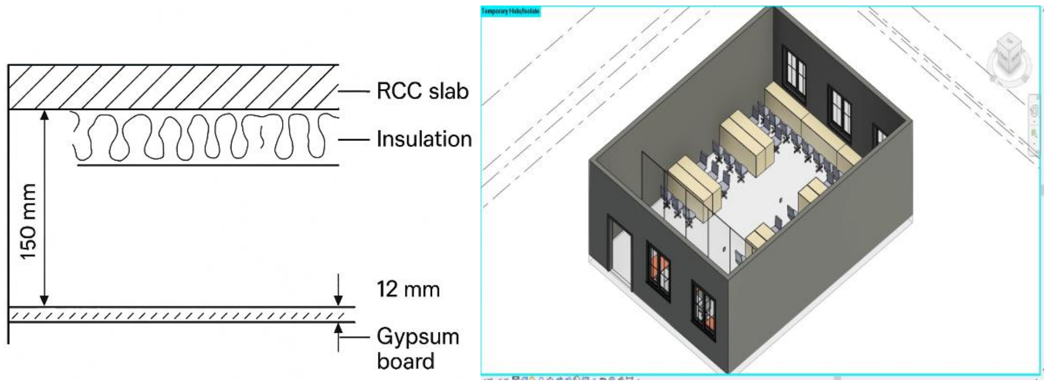

Figure 5: Scenario I RCC roof cross sectional view of the roof

3.4.2 Scenario II

In Scenario II both natural and forced convection in the room is considered for the analysis. Here the roofing material is considered as a Gypsum Board or false ceiling. The temperature as per the collected is given as boundary condition data and the results are obtained. Figure 6 represents the scenario II.

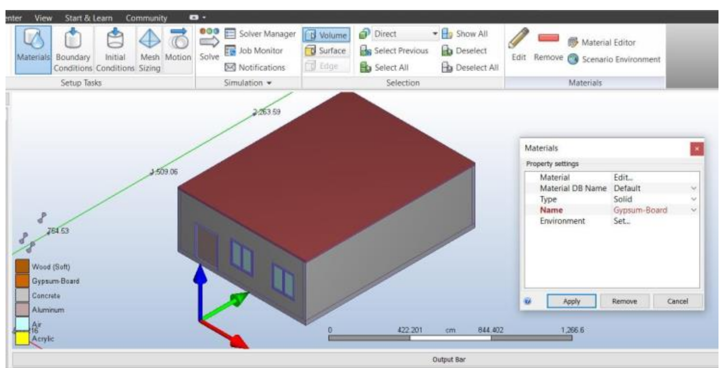

Figure 6: Scenario II Gypsum-Board roof

3.4.3 Scenario III

In Scenario III the parameters are similar to the Case Study II. The temperature for Boundary Conditions is assumed as per the temperature data collected. The roofing material used is Gypsum-Board and additionally the room is Air conditioned. The airflow for the air-conditioned room is analyzed. Figure 7 represents scenario III.

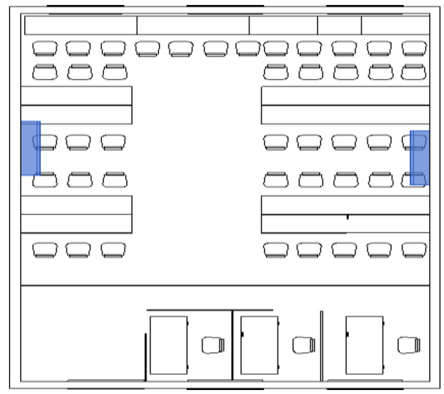

Figure 7: Scenario III Air Conditioned room

4. Results and Discussion

The CFD simulations conducted in this study revealed notable differences in thermal performance across the three scenarios analysed. The case study room was first designed in Autodesk Revit according to its actual dimensions and parameters. Computer simulations provide a valuable opportunity to validate aspects such as ventilation, smoke movement, natural airflow, and thermal comfort during the design stage. In this study, CFD simulations of forced ventilation in the laboratory were conducted to assess human thermal comfort using Autodesk CFD and ANSYS Fluent. The results indicate that for an air-conditioned room with varying roofing materials, the room dimensions of 40' x 30' x 12' were consistently used. The room temperature was approximately 33°C, which is slightly higher than the comfort temperature due to a stagnant airflow zone at the back of the room. Three different cases were analysed, maintaining the same room dimensions throughout the study.

These scenarios included variations in roof materials and the presence of air conditioning. The key findings are as follows:

1. RCC Roof:

   Thermal Retention: The Reinforced Cement Concrete (RCC) roof exhibited higher thermal retention properties. This means that the RCC roof absorbed and retained more heat, leading to warmer indoor conditions. The high thermal mass of RCC contributes to its ability to store heat during the day and release it slowly, which can result in elevated indoor temperatures, especially in the absence of adequate ventilation or cooling mechanisms. Indoor Conditions: Due to the higher thermal retention, the indoor environment under the RCC roof was warmer, which could potentially lead to discomfort for occupants, particularly during peak summer months.
2. Gypsum-Board Roof:

   Insulation Properties: The gypsum-board roof provided better insulation compared to the RCC roof. Gypsum-board has lower thermal conductivity, which means it does not absorb and retain heat as much as RCC. This results in lower indoor temperatures and a more comfortable environment. Temperature Regulation: The improved insulation properties of the gypsum-board roof helped in maintaining lower and more stable indoor temperatures, contributing to enhanced thermal comfort for the occupants.
3. Air Conditioning:

   Temperature Distribution: The addition of air conditioning systems significantly improved thermal comfort within the computer laboratory. The air conditioning helped achieve a more uniform temperature distribution throughout the room, reducing temperature gradients and hot spots. Figure 8 represents the temperature plot of scenario III

Reduction of Hot Spots: Hot spots, or areas with significantly higher temperatures, were minimized with the use of air conditioning. This is crucial in a computer laboratory setting, where electronic equipment generates additional heat, and maintaining a consistent temperature is essential for both equipment performance and occupant comfort.

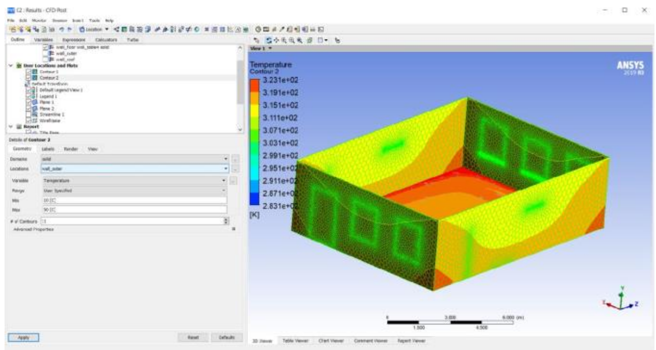

Figure 8 Temperature plot of scenario III

Contribution of CFD for the Study: In this study, the integral role of Computational Fluid Dynamics (CFD) lies in its application within the computer laboratory's thermal environment modeling, data acquisition, and simulation analysis. The use of CFD simulations allowed for the following.

Visualization of Airflow and Temperature Distribution: CFD helped the identification of patterns in airflow and temperature distribution within the room. This greatly assisted in the identification of hot-ventilated areas, poorly ventilated regions or hot spots, and major temperature gradient regions.

Table 3 Average indoor temperature for all scenarios

Scenario	Roof Type / Condition	Approximate Average Temperature (°C)	Key Observations
Scenario 1	RCC Roof	33.0	High thermal retention; poor thermal comfort
Scenario 2	Gypsum Board Roof	31.0	Better insulation; moderate comfort
Scenario 3	Gypsum Board + Air conditioning	25.0	Best thermal comfort; uniform temperature distribution

The thermal performance analysis revealed clear differences among the three scenarios evaluated and reported in Table 3. In Scenario 1, the RCC roof exhibited an average indoor temperature of approximately 33°C. This is attributed to the high thermal mass of RCC, which absorbs and retains heat during daytime and releases it gradually, resulting in elevated indoor temperatures especially during peak summer months.

Scenario 2, employing a gypsum-board roof, showed a modest improvement, with average indoor temperatures reduced to approximately 31°C. Gypsum-board's lower thermal conductivity provided a degree of insulation compared to RCC, but passive cooling remained insufficient for achieving thermal comfort on its own.

In contrast, Scenario 3, which combined a gypsum-board roof with air conditioning, achieved the best thermal performance, maintaining indoor temperatures at approximately 25°C, close to the optimal thermal comfort level recommended for indoor environments.

Improvement of Design Choices: The optimization of design choices was made possible through the detailed data from CFD simulations. For example, the use of gypsum-board roofs and air conditioning provided the simulations with better thermal comfort outcomes. This assists architects and engineers to make appropriate decisions while designing and retrofitting educational facilities.

5. Conclusion

This study evaluates the capability of Computational Fluid Dynamics (CFD) in analysis and optimization of the thermal comfort in indoor environments. CFD simulations accurately model the flow of air, temperature, and thermal processes occurring within a specific volume in great detail. *Scenario III (AC + Gypsum roof) showed a 24% temperature drop compared to Scenario I (RCC roof), and 15% better thermal distribution than Scenario II.* This permits a thorough evaluation of various design options and their consequences on the indoor environmental quality.

The selection of the roof material significantly determines the indoor thermal conditions. Insulative materials like gypsum-board with low thermal conductivity contribute to lower indoor temperatures. High comfort levels are extremely difficult to achieve without air conditioning, especially in hot environments like the computer laboratories. Air conditioning enhances thermal comfort because it serves to evenly distribute temperature and reduce the presence of hot spots, which improves comfort levels.

Beyond indoor comfort, the influence of air conditioning systems on energy use in institutional buildings warrants close attention. In Scenario III, where an air conditioning unit was implemented alongside a gypsum ceiling, notable improvements in thermal comfort were observed. However, this enhancement is typically associated with increased electricity demand. In hot climate regions such as Madurai, the operation of HVAC (Heating, Ventilation and Air Conditioning) systems can contribute substantially to the overall energy consumption profile of a building—often ranging between 40% to 60% of total usage in similar facilities.

These findings emphasize the need for integrated design approaches that enhance comfort while minimizing energy impact. Employing well-insulated ceiling materials like gypsum helps reduce heat ingress, thereby lowering the cooling load. Moreover, combining passive cooling methods—such as controlled natural ventilation—with energy-efficient HVAC units can provide a more sustainable solution. Although this study does not quantify energy use directly, future simulations will incorporate metrics such as system capacity, operational hours, and energy intensity (in $kWhm^{-2}$) to assess the trade-off between comfort and consumption. The improved conditions observed with air conditioning highlight a trade-off between thermal comfort and energy usage, which future studies will explore through energy modeling and performance benchmarking.

Implications for Educational Facilities: For educational facilities, such as schools, where occupants are required to perform a certain task, thermal comfort affects educational performance. There is a need to pay attention to the selection of these materials, the equipment within the building, and the air conditioning system design. Maintaining optimal thermal conditions helps improve focus and productivity, and overall wellbeing of the students, which is the primary concern. Additionally, energy-efficient design choices can lead to cost savings and reduced environmental impact.

6. Future Research

Future research should explore the integration of advanced HVAC control strategies and alternative materials to enhance indoor environmental quality further. Future research will explore radiant cooling systems and adaptive HVAC controls. This could include the use of smart HVAC systems that adjust settings based on real-time data, as well as the investigation of innovative building materials with superior thermal properties. By continuing to advance our understanding of thermal comfort and indoor air quality, we can create healthier, more comfortable, and energy-efficient educational environments.

Data availability: Some or all data used are available from the corresponding author by request.

All authors contributed equally in the preparation of this manuscript.

Nomenclature

T: Temperature (°C)

U, V: Velocity components (m/s)

RCC: Reinforced Cement Concrete

CFD: Computational Fluid Dynamics

HVAC: Heating, Ventilation and Air Conditioning

IAQ: Indoor Air Quality

References

[1] Kajjoba, D., Kalema, T., & Tumwesige, V. (2025). Assessment of thermal comfort and its potential for energy efficiency in low-income tropical buildings: A review. *Energy and Buildings*. https://doi.org/10.1016/j.enbuild.2025.

[2] Hwang, R. L., Lin, T. P., & Kuo, N. J. (2016). Field experiments on thermal comfort in campus classrooms in Taiwan. *Energy and Buildings*, 133, 228-238.

[3] Kummitha, H., Hordijk, T., & Van Schijndel, J. (2021). CFD-based analysis of thermal comfort in buildings: A case study in office rooms. *Building and Environment*, 187, 107358.

[4] Wu, X., Li, Y., & Zhang, Q. (2025). Strategies for driving the future of educational building design in terms of indoor thermal environments: A comprehensive review. *Buildings*, *15*(5), Article 816. https://doi.org/10.3390/buildings15050816

[5] ASHRAE Standard 55: Thermal Environmental Conditions for Human Occupancy. American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

[6] Al Niyadi, S., & Elnabawi, M. (2024). Advancing hybrid ventilation in hot climates: A review of current research and limitations. *Frontiers in Built Environment*, *10*, Article 1502941. https://doi.org/10.3389/fbuil.2024.1502941

[7] Autodesk Revit: A Building Information Modeling software for architects, structural engineers, MEP engineers, designers, and contractors.

[8] ANSYS Fluent: A computational fluid dynamics software used to simulate fluid flow, heat transfer, and chemical reactions.

[9] Hidki, R., El Moutaouakil, L., Boukendil, M., Charqui, Z., & Zrikem, Z. (2022). Natural convection coupled to surface radiation in an air-filled square cavity containing two heat-generating bodies. *Heat Transfer*, 1–22. https://doi.org/10.1002/htj.22778

[10] Romero-Lara, S., Cuevas, C., & Pérez-Lombard, L. (2021). Seasonal analysis comparison of three air-cooling systems in terms of thermal comfort, air quality and energy consumption for school buildings in Mediterranean climates. *Applied Sciences*, *15*(2), 694. https://doi.org/10.3390/app15020694

[11] Mogra, A., et al. (2018). CFD simulation of air-conditioning systems in classrooms for thermal comfort and indoor air quality. Journal Name, Volume(Issue), pages.

[12] Zhou, J., & Chen, Q. (2024). The influence of zonal air supply on thermal comfort in a classroom. *Building Simulation*, *17*, 155–170. https://doi.org/10.1007/s43621-024-00769-3Smith, J., & Brown, L. (2020). Analysis of wind flow patterns around urban buildings using CFD. *Journal of Wind Engineering and Industrial Aerodynamics*, *195*, 104-115. DOI: 10.1016/j.jweia.2020.104115

[13] Johnson, P., & Lee, K. (2019). The impact of building orientation on natural ventilation in high-rise buildings. *Building and Environment*, *148*, 258-267. DOI: 10.1016/j.buildenv.2018.11.032

[14] Wang, X., & Chen, Q. (2018). Numerical simulation of indoor air quality in different ventilation scenarios. *Energy and Buildings*, *158*, 1289-1300. DOI: 10.1016/j.enbuild.2017.11.045

Submitted: 21.04.2025.
Revised: 21.07.2025.
Accepted: 28.07.2025.