

Computational Fluid Dynamics Analysis of Air Circulation and Thermal Comfort in a Lecture Room

Ekemini Joseph ¹, Bori Ige ², Adamu Shanono ³

¹ Department of Mechanical Engineering, Baze University, Abuja – Nigeria

Email: ekeminijoseph2024@gmail.com

² Department of Mechanical Engineering, Federal University of Technology, Minna – Nigeria

Email: ige.bori@futminna.edu.ng

³ Department of Mechanical Engineering, Baze University, Abuja – Nigeria

* Corresponding author: adamushanono@gmail.com

ARTICLE INFO

Keywords:

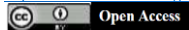
Lecture Room,
Air Conditioning,
Thermal Comfort,
CFD,
Simulation.

Vol.01, No.04, 2026

<https://doi.org/10.5281/zenodo.22332658>

Copyright © 2026 by author(s) and Scientific Research Publishing Inc. This work is licensed under the Creative Commons Attribution International License (CCBY4.0).

<http://creativecommons.org/licenses/by/4.0/>



ABSTRACT

This study presents a Computational Fluid Dynamics (CFD) analysis of air circulation and thermal comfort in Lecture Room D58, Faculty of Engineering, Baze University, Abuja - Nigeria. The purpose of the research was to evaluate the effect of Air Conditioning (AC) units' placement on airflow distribution and temperature reduction. This was in order to determine the best possible locations of the AC systems for effective indoor thermal comfort in the lecture room. A three-dimensional model of the lecture room was developed using ANSYS Design Modeler, and six different AC placement configurations were simulated in ANSYS Fluent. The simulations solved the Navier–Stokes, continuity, and energy equations using appropriate turbulence models to predict airflow velocity and temperature distribution. The number of AC units (two and four units) and their vertical positions (2.8 m and 3.4 m above the floor) were varied in the six cases. Results showed that the initial room temperature of 33°C was reduced differently across the six cases. Case 2 on the YZ plane with two air conditioners positioned 2.8 m from the floor achieved the highest temperature reduction, lowering the room temperature from 33°C to 17°C (a 16°C decrease). Similarly, air velocity distribution in Case 2 was more uniform across the occupied zone compared to other cases. On the other hand, Case 1 on the YZ plane with two air conditioners positioned at 3.4 m from the floor and Case 6 on the XY plane positioned also at 3.4 m from the floor showed poor circulation with limited temperature reduction. Meanwhile, the cases with four AC units did not produce proportional improvement in comfort despite increased airflow. The study demonstrated that AC placement and inlet height are more critical than the number of units for effective cooling. It also provided insights into optimizing interior air quality and thermal comfort for lecture environments by ensuring adequate space between the AC units and top of the room.

1. Introduction

Lecture Room D58 is one of the lecture venues in the faculty of engineering at Baze University, Abuja – Nigeria. The lecture venue serves four departments of the faculty at different times. In the past three years, several complaints were received from lecturers and students on the lack of effective cooling from the air conditioners located in the lecture room. This necessitated this study considering that the two Samsung split units air-conditioning systems of 2 horsepower each currently in the lecture room, were serviced and certified in good condition. Consequently, Computational Fluid Dynamics (CFD) modelling was utilised to tackle the inadequate ventilation and thermal comfort in the lecture room.

According to Anderson (1995), the architecture and engineering sectors have seen a substantial change toward favouring sustainable and occupant-centric solutions in recent years. This progression emphasizes the significance of achieving a balance between energy economy and human comfort, especially in enclosed settings. As a result, there is an increasing demand for a thorough analysis of air circulation and thermal comfort dynamics in indoor environments.

Previous studies highlighted airflow principles and thermal comfort metrics (Awbi, 2003; De Dear & Brager, 2013). Thermal comfort is defined by air temperature, humidity, movement, and preferences

(De Dear & Brager, 2013), with effective circulation reducing gradients (Ghogare et al., 2020). Anderson (1995) emphasized CFD's role in simulating indoor environments by using airflow principles rooted in fluid dynamics, distinguishing laminar and turbulent flows, while Chen (2009) and Liu et al. (2019) demonstrated its application in ventilation optimization.

Launder & Spalding (1974), Wilcox (2006) and Chen (2009) noted that ANSYS Fluent is a computational fluid dynamics software that simulates fluid flow using Navier-Stokes equations. These equations describe the motion of viscous fluids and are expressed in differential form. ANSYS Fluent excels in simulating indoor airflow, solving Navier-Stokes, continuity, and energy equations, alongside turbulence models like $k-\epsilon$ and $k-\omega$.

White, (2011) asserted that continuity equation expresses the principle of mass conservation in fluid flow, stating that the rate of change of mass density within a fluid element is equal to the negative divergence of the mass flux vector. In simpler terms, it means that the amount of fluid entering or leaving a region must equal the rate of change of fluid density within that region.

Munson et al. (2012) emphasised that Navier-Stokes or momentum equations describe the conservation of momentum in fluid flow, accounting for the effects of pressure, viscous forces, and external body forces on the fluid motion. They indicated that the speed and direction of fluid flow evolve under the influence of different forces. Cengel and Ghaja (2014) further noted that energy equation governs the transfer of thermal energy within a fluid, considering mechanisms such as conduction, convection, and heat sources or sink. In simpler terms, it tells us how heat moves through a fluid and affects its temperature.

Thus, ANSYS allows users to model, analyse, and visualize complex airflow dynamics which is critical in connecting theoretical knowledge with practical implementations, resulting in improved indoor air circulation. It was used to undertake a thorough analysis of air circulation and thermal comfort in lecture room D58. ANSYS Fluent, a component of the software suite, was used to create detailed room models, analyse airflow patterns, and anticipate thermal comfort levels for occupants.

Consequently, six AC placement cases excluding doors and windows were evolved for the study, each scenario involved the relocation of the AC units as necessary, with an emphasis on assessing velocity and temperature distribution through CFD simulation analysis. The height of the lecture room is 4 m and has normal capacity for 40 students while the maximum capacity is 50 students. Two AC units are currently in the room each located at 3.4 m from the floor and the distance between the two AC units is 2.4 m. The current location of the AC units in the lecture venue is represented as Case 1.

2.0 Materials and Methods

2.1 Materials

The materials utilised to conduct the research were a thermometer and CFD software.

Thermometer: A dry and wet bulb thermometer was used to measure the ambient temperature within the study area as shown on figure 1.



Figure 1: Analogue Dry and Wet Thermometer.

Computational Fluid Dynamics Software: ANSYS Fluent was utilised for 3D airflow and heat transfer analysis. Its structure includes pre-processor (geometry and mesh), solver (numerical computations), and post-processor (visualization).

2.2 Methods

The study employed CFD to assess air circulation and thermal comfort in Room D58. The methodology included;

Study Design: The use of ANSYS Fluent software tool for fluid flow analysis and heat transfer modelling. This structured approach within the ANSYS Workbench environment streamlines the CFD simulation process, from geometry generation to result visualization and reporting, allowing for rapid and precise investigation of fluid dynamics phenomena. Figure 2 shows the ANSYS Fluent CFD algorithm.

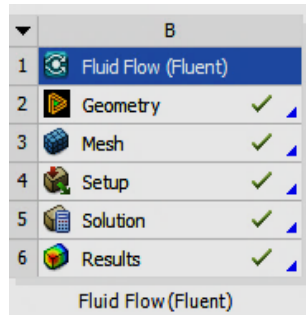


Figure 2: ANSYS Fluent CFD Algorithm.

Geometric Modelling: A 3D model was developed in ANSYS Fluent geometry, incorporating walls, windows, doors, and AC units. To achieve a perfect 3D build for the model, features such as the room walls, windows, doors, and other extras were created using a variety of functions and processes such as 2D sketching and dimensions, 3D executions such as extrusion of frozen material, add material, slice material, cut material, and chamfer.

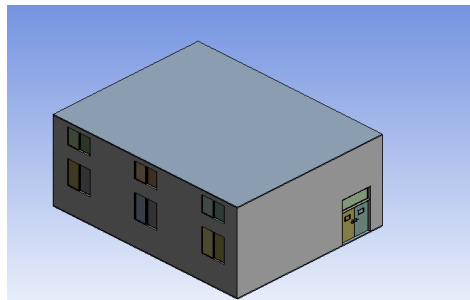


Figure 3: Outer Geometry of Room D58.

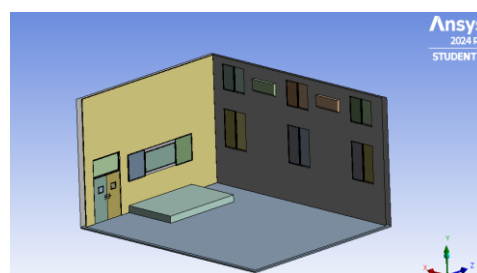


Figure 4: Inner Geometry of Room D58.

Figure 3 shows the outer geometry of room D58, while figure 4 shows the inner geometry of the classroom. As stated above, since the ANSYS software uses 3D XYZ coordinate system, the labels YZ and XY were assigned to indicate the 2D plane projections used in analysing the 3D model.

Scenarios: Six cases were postulated for the study as follows;

Case 1: 2 ACs on YZ plane at 3.4 m from the room floor.

Case 2: 2 ACs on YZ plane at 2.8 m from the floor.

Case 3: 4 ACs on YZ and XY planes at 3.4 m from the floor area.

Case 4: 4 ACs on YZ and XY planes at 2.8 m from room floor.

Case 5: 2 ACs on XY plane at 2.8 m from the floor area.

Case 6: 2 ACs on XY plane at 3.4 m from the room floor.

Mesh Generation: CFD simulations rely heavily on mesh generation, also known as computational domain discretization. It entails splitting the physical domain into smaller control volumes or controllable parts in order to numerically solve the governing equations. Mesh quality has substantial impact on the accuracy of results. Figure 5 shows the mesh generation that discretised the lecture geometry. The geometry was developed as a representative of the lecture room model.

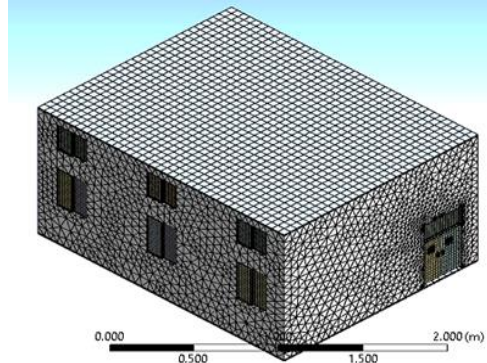


Figure 5: The Lecture Room Mesh Geometry.

2.21 Simulation Setup: The setup process includes solving equations to achieve desired results, considering parameters such as gravity, temperature, flow type, differential equations, boundary conditions (Dhakar & Patel, 2018), and k-ε turbulence models. The setup toolbox is shown in figure 6.

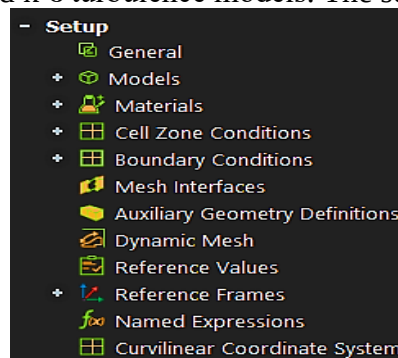


Figure 6: Setup Toolbox.

The working materials selected were air and concrete. On selection of the working materials for AC system modelling, the material properties such air density, viscosity, specific heat and thermal conductivity were automatically assigned from the ANSYS Fluent material database under standard atmospheric conditions. The solver computes the corresponding outlet flow parameters based on the governing equations for mass, momentum, and energy.

2.22 Airflow Analysis: Post-processing tools in ANSYS Fluent were used to show airflow velocity vectors, streamlines, and contours.

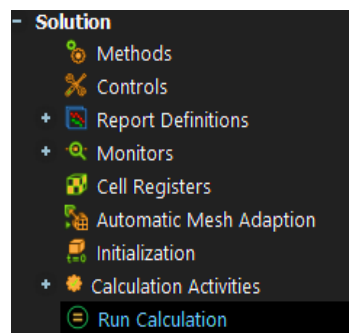


Figure 7: Solution Tool Box.

Running Simulations: The CFD solver solved the governing equations shown below by modelling airflow, temperature distribution, and other variables based on the number of iterations.

a. Continuity Equation (Conservation of Mass):

$$\frac{\partial}{\partial t}(\rho) + \frac{\partial u}{\partial x}(\rho) + \frac{\partial v}{\partial y}(\rho) + \frac{\partial w}{\partial z}(\rho) = 0$$

b. Momentum Equations (Navier-Stokes equations):

Equation for velocity at X-Direction:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = F_u - \frac{1}{\rho} \cdot \frac{\partial P}{\partial x} + \nu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$

Equation for velocity at Y-Direction:

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = F_v - \frac{1}{\rho} \cdot \frac{\partial P}{\partial y} + \nu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right]$$

Equation for velocity at Z-Direction:

$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = F_w - \frac{1}{\rho} \cdot \frac{\partial P}{\partial z} + \nu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right]$$

c. Energy Equation (for Thermal Simulations):

$$\rho C_p \left[\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right] = Q + k \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right]$$

3.0 Results and Discussion

Case 1: 2 AC Units on YZ Plane 3.4 m From the Floor.

Figure 8 illustrates the airflow velocity distribution in room D58 for Case 1. Air velocity was concentrated near the ceiling, with weaker airflow in the occupied zone, indicating uneven air circulation.

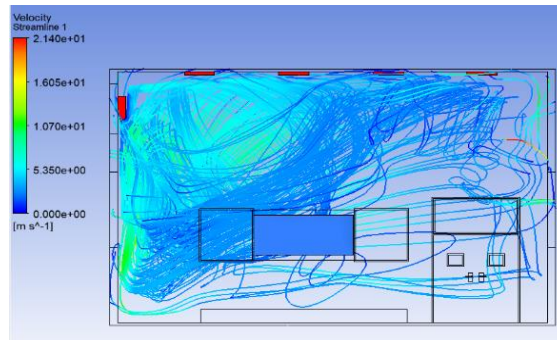


Figure 8: Streamline Velocity at YZ Plane

Figure 9 is the chart of temperature against Velocity for Case 1. It can be seen that as air velocity increased, the temperature decreased, showing that higher airflow improves heat removal within the room. The dominant temperature value observed in the temperature-velocity chart was 26.15°C, indicating moderate cooling effectiveness with increase in velocity.

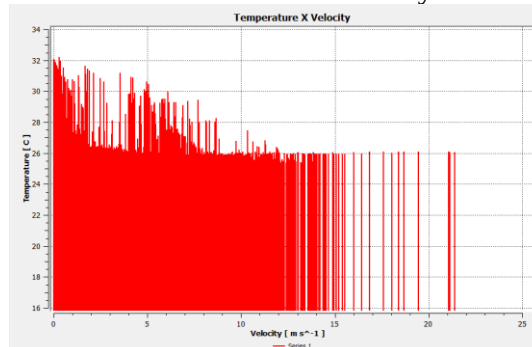


Figure 9: Chart of Temperature against Velocity for Case 1.

CASE 2: 2 AC Units placed 2.8 m From Floor Area.

Figure 10 shows Streamline Velocity at YZ Plane for Case 2. There was more even distribution of air in the lecture venue compared to Case 1.

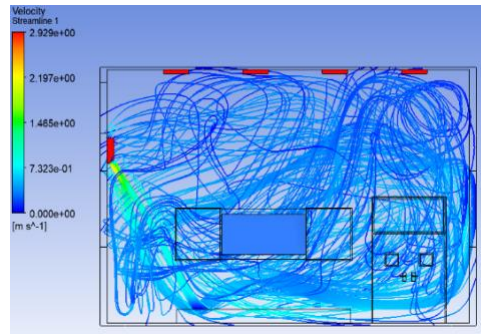


Figure 10: Streamline Velocity at YZ Plane.

Similarly, Figure 11 is the chart of temperature against velocity for case 2. There was noticeable reduction in temperature throughout the room with the dominant temperature value observed in the temperature-velocity chart being 17°C. There was thus effective cooling with more uniform temperature distribution than in Case 1.

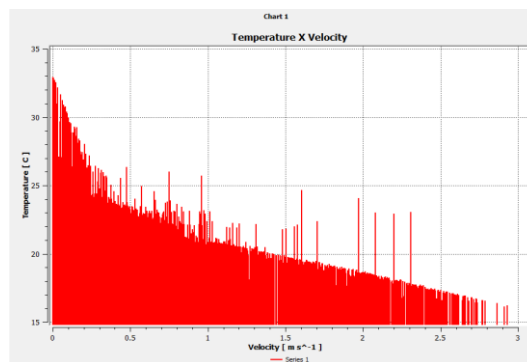


Figure 11: Chart of Temperature against Velocity.

CASE 3: 4 AC Units on YZ and XY Planes Placed 3.4 m from Floor Area.

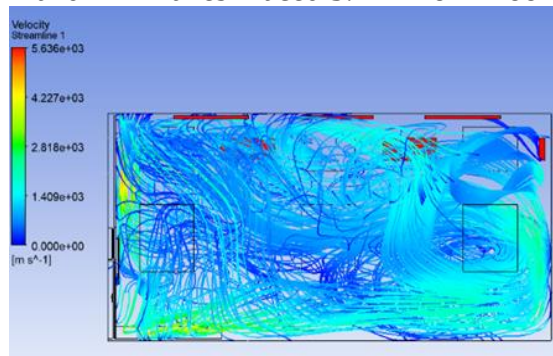


Figure 12: Streamline Velocity for Case 3.

Figure 12 shows higher airflow velocities compared to Cases 1 and 2, with streamlines concentrated near AC inlets. While this arrangement increased airflow in the lecture room, occupied zones still experienced uneven flow.

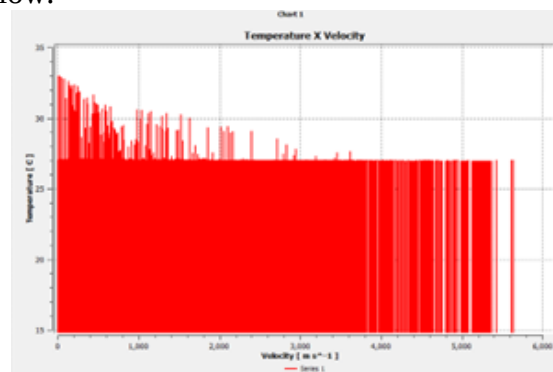


Figure 13: Chart of Temperature against Velocity for Case 3.

Meanwhile, figure 13 shows the chart of temperature against velocity for Case 3. From the figure, despite high air velocities, temperature reduction was limited with minimum temperatures around 27.2°C, suggesting that increased airflow alone does not ensure improved thermal comfort.

CASE 4: 4 AC Units on YZ and XY Planes placed 2.8 m from Floor Area.

Figure 14 depicts Streamline Velocity at YZ Plane of four AC units positioned with a floor height of 2.8 m. This configuration was tested to evaluate the effect of lower AC placement.

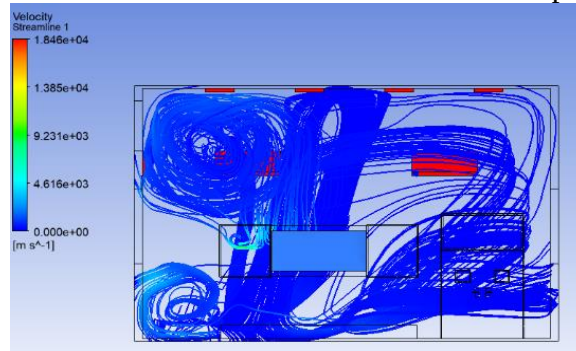


Figure 14: Streamline Velocity for Case 4.

However, in spite of high velocities near inlets, airflow in the occupied zones remained uneven, suggesting that lower inlet placement alone does not guarantee improved circulation. Figure 15 shows the chart of temperature against velocity for Case 4. There was minimal temperature reduction, with average values around 26.8°C, indicating limited cooling effectiveness.

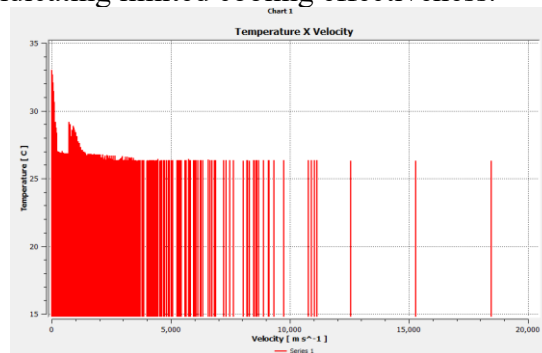


Figure 15: Chart of Temperature against Velocity for Case 4.

CASE 5: 2 AC Units on XY Plane placed at 2.8 m from Floor.

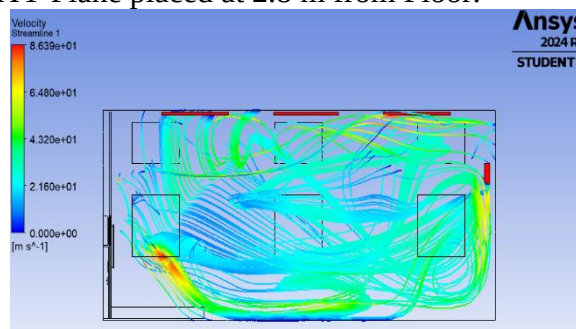


Figure 16: Streamline Velocity at XY Plane

Figure 16 indicates moderate airflow near AC units but weaker circulation in occupied zones.

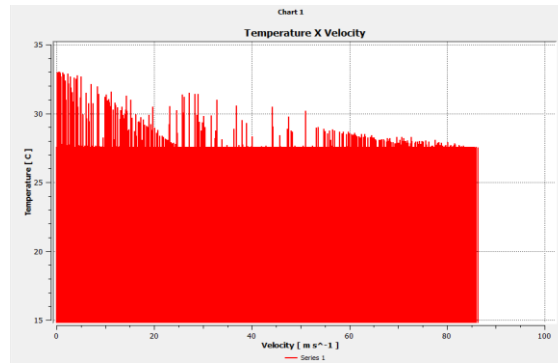


Figure 17: Chart of Temperature against Velocity for Case 5.

From figure 17, the dominant temperature value observed in the temperature-velocity chart was 27.8°C, which reflects inadequate thermal comfort as the temperature reduction was minimal despite moderate airflow.

CASE 6: 2 AC Units on XY Plane placed at 3.4 m from Floor Area.

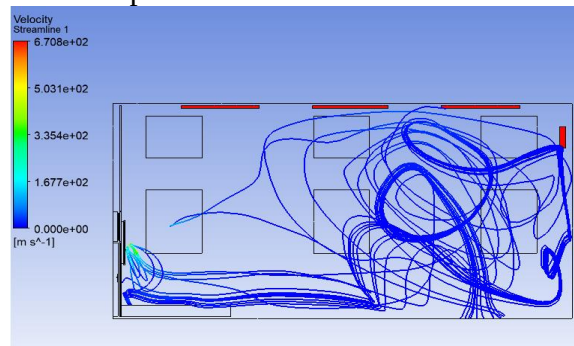


Figure 18: Streamline Velocity at XY Plane.

Figure 18 shows concentrated airflow near AC units but weak circulation throughout occupied areas. Similarly, Figure 19 indicates limited temperature reduction, with average temperatures around 27.6°C. Thus, the temperature remained relatively high, showing poor cooling effect.

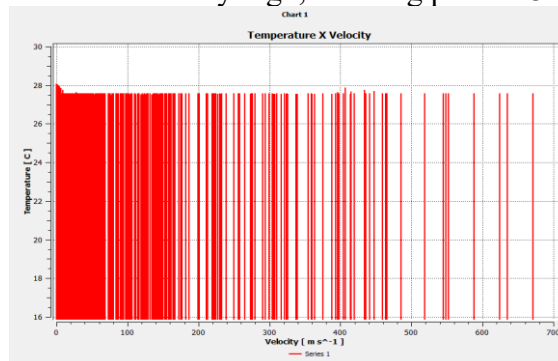


Figure 19: Chart of Temperature against Velocity for Case 6.

4.0 Conclusion

This study investigated air circulation and thermal comfort in room D58 of the Faculty of Engineering, Baze University, using ANSYS-based computational fluid dynamics (CFD) simulations across six different AC placement scenarios. Case 2 on the YZ plane with two air conditioners positioned 2.8 m from the floor achieved the highest temperature reduction, lowering the room temperature from 33°C to 17°C (a 16°C decrease). Similarly, air velocity distribution in Case 2 was more uniform across the occupied zone compared to other cases. On the other hand, Case 1 on the YZ plane with two air conditioners positioned at 3.4 m from the floor and Case 6 on the XY plane positioned also at 3.4 m from the floor showed poor circulation with limited temperature reduction. Meanwhile, the cases with four AC units did not produce proportional improvement in comfort despite increased airflow.

The study demonstrated that AC placement and inlet height are more critical than the number of units for effective cooling. The findings can guide future improvements in HVAC design and interior

layouts to enhance occupant comfort, health, and productivity. The results also provided insights into optimizing interior air quality and thermal comfort for lecture environments by ensuring adequate space between the AC units and top of the room.

Acknowledgements

The authors specifically thank the Dean, Faculty of Engineering at Baze University, Abuja for giving the authors the opportunity to undertake this study. The assistance and cooperation of the faculty technical staff is also acknowledged.

Author Contributions

All the authors contributed to this research study. The first author was responsible for initiating the idea of the study as part of his degree project and the collation of data. He also wrote the original draft of the manuscript. The second author gave direction on the methodology, while the third author reviewed and edited the manuscript.

Funding

The authors did not receive any kind of financial support for the conduct of the research, preparation and writing of the article, and; collection, analysis and interpretation of data.

Conflict of Interests

The authors declare no conflicts of interest in the preparation and writing of this paper.

References

1. Anderson, J.D. (1995). Computational Fluid Dynamics: Principles and Applications. McGraw-Hill, New York. 547 pp.
2. ANSYS Inc. (2023). ANSYS Fluent Theory Guide. ANSYS Inc., Canonsburg, PA. Available at: <http://www.ansys.com>. Retrieved 15 January 2024.
3. ASHRAE. (2023). ASHRAE Handbook – Fundamentals. American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, GA.
4. Awbi, H.B. (2003). Ventilation of Buildings. CRC Press, London. 522 pp.
5. Çengel, Y.A. and Ghajar, A.J. (2014). Heat and Mass Transfer: Fundamentals and Applications (5th ed.). McGraw-Hill Education, New York. 906 pp.
6. Chen, Z. (2009). Computational Fluid Dynamics: An Introduction. Elsevier Science, Oxford. 312 pp.
7. Dhakar, P. S. and Patel, A. (2018). CFD Analysis of Air Conditioning in a Room using ANSYS Fluent. Journal of Engineering and Technology, 10(2), 45-58.
8. De Dear, R. and Brager, G. (2013). Thermal comfort in naturally ventilated buildings: Revisions to ASHRAE Standard 55. ASHRAE Transactions, 109(2): 1–13.
9. Ghogare, A.G., Sinha, S.L. and Verma, T.N. (2020). Effect of inlet airflow direction on the indoor environment of a naturally ventilated room using CFD. Journal of Building Engineering, 32: 101–116.
10. Launder, B.E. and Spalding, D.B. (1974). The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3(2): 269–289.
11. Liu, J., Zhu, S. and Kim, M.K. (2019). A review of CFD analysis methods for personalized ventilation in indoor built environments. Indoor Air, 29(1): 5–7.
12. Munson, B.R., Young, D.F., Okiishi, T.H. and Huebsch, W.W. (2012). Fundamentals of Fluid Mechanics (7th ed.). John Wiley & Sons, Hoboken, NJ. 792 pp.
13. Nielsen, P.V. and Buus, M. (2009). Indoor Environment: A Sustainable and Health Perspective. CRC Press, Boca Raton, FL. 361 pp.
14. White, F.M. (2011). Fluid Mechanics (7th ed.). McGraw-Hill Education, New York. 862 pp.
15. Wilcox, D.C. (2006). Turbulence Modeling for CFD (3rd ed.). DCW Industries, La Cañada, CA. 522 pp.