Design and Simulation of Air Conditioning System in a Large Auditorium Based on Computational Fluid Dynamics

R. Abu, K. A. Oladejo, A. O. Popoola, K. T. Oriolowo and K. M. Odunfa

Abstract—Air conditioning system is an indispensable part of buildings today. The cost of this system increases with the rise in energy consumption which poses a challenge as well as air distribution in large auditoria. Analysis of results can also be daunting when designing this system. The study focuses on designing an air conditioning system in a large auditorium, applying computational fluid dynamics and visualizing the result in a Virtual Reality (VR) environment. The 3-dimensional model of the 520-capacity Technology Lecture Theatre, University of Ibadan, Nigeria was drawn with Autodesk Revit and modified into the geometry applicable for Displacement Ventilation (DV) and Mixed Ventilation (MV) for ease of numerical analysis with ANSYS Fluent. The building model and simulation results were then imported into Unity software for visualization in VR. The DV achieved better thermal comfort and air distribution in the computer simulation. At a supply temperature of 292.15 K, the DV system was able to keep the auditorium temperature at about 296.50 K, while the MV system at a supply temperature of 289.15 K was only able to maintain the temperature at 295.40 K. The temperature profile showed that the lower region where the students were seated was colder in DV compared to MV by at least 3 K. The results were also observed from a convenient position in VR. This study, with the aid of computational fluid dynamics and virtual reality, was able to establish that displacement ventilation design has better airflow, lower energy consumption and is efficient for an air conditioning system in a large auditorium.

Index Terms—Computational Fluid Dynamics, Displacement Ventilation, Mixed Ventilation, Virtual Reality.

I. INTRODUCTION

Air Conditioning (AC) system removes heat and moisture from the interior of an occupied space to improve the comfort of occupants. It uses a fan to distribute the conditioned air in the occupied space, such as a building to improve thermal comfort and indoor air quality. Almost all regions around the world use air conditioning systems of various types in their indoor environment. People, who work in buildings or occupied spaces, release moisture and emit heat during work. These moisture and heat must be considered when determining air conditioning requirements, hence the constant need to improve the efficiency of these systems. There are vital factors that must be considered in order to ensure efficiency of the system such as thermal comfort, airflow, air purity, temperature, humidity and energy efficiency.

The minimal energy production in Nigeria has failed to meet the demand of the populace. People want more for their day-to-day businesses and activities. Cool and clean air is one of the significant needs of these people. However, the energy consumption and cost of air conditioning systems are not within their capacity.

Various forms of cool and clean air distribution have been studied extensively over the years, and the researches have been able to show through experiments and simulations, the advantages and disadvantages of the different forms [1]. Air conditioning in large spaces consumes much energy. As a result, different researches and analyses were carried out varying different parameters and optimizing for better energy efficiency. Hua et al. [2] analyzed energy saving through numerical simulation. In the research, the existing Constant Air Volume (CAV) compared with a proposed Variable Air Volume (VAV) air conditioning system of which the latter meets the requirement for thermal comfort with higher exergy and energy efficiencies.

Two different AC air distribution techniques were explored in this study, mixed ventilation and displacement ventilation. Mixed ventilation technique is more common and has been around for years. This technique involves setting up the supply diffusers in the ceiling while the returns are close to the floor of the room. Stagnation of air and high-temperature concentration are challenges of this technique [3]. This stagnation affects the air quality and load on the AC system, hence, higher energy consumption.

Gilani et al. [4] had shown that using the Displacement Ventilation (DV) technique in large auditoria with a high ceiling over mixing air systems to be more cost-effective and of better thermal comfort for the occupants. This technique is based on the layer stratification concept in which the dense cold air from the diffuser is delivered at a lower level, the cold air on absorbing heat from the occupants and other internal heat sources become warm and lighter. This warm air rises above the cold air carrying with it the impurities in the air and exiting the room through the...
exhaust grilles on the ceiling. It has been found out from this study that implementing secondary air return system and increasing the area of the supply inlets aid in maximizing energy consumption and prevention of thermal discomfort [1]. Displacement ventilation is not only useful in cooling large spaces, with more analysis, but it is also capable of providing satisfactory thermal comfort for the occupants [5].

In this present study, a numerical analysis of the airflow was carried out using Computational Fluid Dynamics (CFD) to investigate the flow pattern of each ventilation technique. CFD is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. The advantages are helpful to obtain insight into problems where an analytical solution can be intimidating or experimental measurements are challenging to get. In past studies, CFD has been employed in the design of air conditioning systems for large auditoria, these studies have shown how versatile CFD software are at simulating air distribution and flow pattern to help in designing better systems [6]–[10]. Kwong et al. [5] studied computational simulation in a tropical region and found out that the simulation results were convincing and reliable. The results were also consistent with the outcome of their experiment even though there were a few discrepancies, the overall results correlated. Won et al. [11] successfully modelled spatial distribution of gas-phase species in indoor conditions using CFD.

Building simulation helps to ensure adequate communication between the engineer and client which lead to better customer satisfaction. The CFD software delivers acceptable analysis with high accuracy depending on how detailed the meshes are set. One of the significant limitations experienced in this area is insufficient computing power to apply finer meshes to get very accurate results [3]. Suitable boundary conditions need to be applied in the numerical analysis of the CFD. Analyzing the effects of these boundary conditions on the space being considered is an effective way to optimize the AC design.

Up to the time of development of CFD, the design of HVAC systems was very stressful and unexciting. The prediction of the flow of air and temperature fields were based on empirical characteristics and experimental results gotten from laboratory studies on the behaviour of various AC components [12]. The ANSYS Fluent is a primary CFD simulation tool that provides functionalities for pre-processing, analysis and calculation based on the set parameters and boundary conditions, also, post-processing which aids integration with other tools, such as visualization software [1].

Realistic visualization of CFD results had been improved by using Virtual Reality (VR) [13]–[14] and Augmented Reality (AR) [15]–[17] applications. Juan et al. [18] formulated research agenda for VR and AR uses in architecture, engineering and construction.

Virtual Reality (VR) is a 3D image produced by a computer or imagery of an environment that people can interact with by using sensors and specialty devices, such as a built-in screen helmet or glove-wearing seemingly real or physical to the person. VR offers a more immersive experience in observing computer graphics which is quite useful in engineering, especially in manufacturing. A VR application was developed in the course of this study for personal computer and android devices, especially for users to experience the virtual environment on their respective devices.

One of the technological advances made in recent year is in virtual reality. Visualization of simulation results is just one of the applications of this technology to engineering. The results of the CFD can be exported into Unity software to make for a better visualization of ANSYS Fluent results. Unity software provides the platform to develop for a virtual reality environment. In some previous studies, integration of CFD into Virtual Reality environment has led to better understanding of the CFD results by non-engineering disciplines such as architects, designers and even the non-professionals in general to make informed decisions [19].

Visualization is of high importance in constructing environment-minded buildings which aim at providing thermal comfort using minimal energy with reduced negative impact on the environment [13]. The author pointed out how previous researches encountered some problems for example, in the data imported into Unity, information on colour in temperature results and arrow in wind flow were not kept. Also, from these previous studies, researchers had to manually go back into the CFD software to tweak parameters in order to confront any unpleasant results visualized in Unity. This problem of no immediate feedback can cause delay in design processes.

The output of visualization leverages on new technologies in order to enable the viewers to experience vivid and immersive interaction with numerical simulation results [20].

Therefore, this study investigates the problem of high energy consumption, air quality, and distribution of the air conditioning system. In addition, the visualization of the CFD simulations is developed.

II. METHODOLOGY

The procedure for the analysis of the air conditioning follows two paths, mixed ventilation systems which is the existing design and displacement ventilation as the proposed design.

A. Building Model

The building under consideration is the Technology Lecture Theatre (TLT), University of Ibadan, Ibadan, Nigeria. The medium-sized auditorium is shown in Fig. 1.

Fig. 1. Technology Lecture Theatre

DOI: http://dx.doi.org/10.24018/ejers.2020.5.9.2062
The 3D model of the building was created using Autodesk Revit software which is shown in Fig. 2.

![Autodesk Revit 3D Model](image)

The existing model of the auditorium has an ineffective air distribution at full capacity of 520 students. This brought about the study to design a new ventilation system for the auditorium. Before getting into the distinctions of the present and proposed ventilation system, the design of each ventilation system is based on the calculated cooling load capacity of the auditorium.

### B. Total Cooling Load Calculation

The design conditions for this study are shown Table I.

#### TABLE I: DESIGN CONDITIONS

| Condition          | Design Condition               |
|--------------------|--------------------------------|
| Building           | Technology Lecture Theatre     |
| Location           | Ibadan, Nigeria                |
| Elevation          | 227.00 m                       |
| Average wind velocity | Low                           |
| Wind Direction     | South-West                     |
| Design Month       | March                          |
| Outdoor Dry Bulb Temperature | 35°C                          |
| Outdoor Wet Bulb Temperature | 27.1°C                        |
| Indoor Dry Bulb Temperature | 25.5°C                        |
| Daily Range        | 11.6°C                         |
| Relative Humidity  | 55%                            |
| Latitude           | 7°26'28"                       |
| Longitude          | 3°53'34"                       |
| Air Velocity       | 48 fpm                         |
| Air Requirement    | 0.0071 m³/s/person             |
| Estimated Occupancy| 522 persons                    |
| Expected Occupants | Students and Lecturer          |

The cooling load calculation was done through two of the three methods documented in ASHRAE Handbook Fundamentals [21]:

i. Transfer Function Method (TFM).

ii. Cooling Load Temperature Difference / Cooling Load Factors (CLTD/CLF).

### C. Cooling Load Temperature Difference / Cooling Load Factors

This method is derived from TFM and uses tabulated data to simplify the calculation process. The method can be relatively quickly transferred into simple spreadsheet programs but has some limitations due to the use of tabulated data.

1) **Sensible Heat Gain:**
   - Roofs – 5,996.07 W
   - Walls – 7,133.77 W
   - Window – 7, 586.82 W
   - Partitions, ceilings and floors – 10,703.47 W
   - People - 31,826.34 W
   - Lighting – 1,895.40 W
   - Infiltration – 2, 429.60 W
   - Ventilation – 2, 429.60 W
   - The total cooling sensible load is 70,001.07 W.

2) **Latent Heat Gain:**
   - The latent heat is due to the people in the auditorium (the students).
   - People – 23,490.00 W

3) **Peak Load:**
   - Summing up the cooling sensible and latent load, we have a peak load of 93,491.07 W.

### D. Transfer Function Method

This method requires the use of a computer program or an advanced spreadsheet. It is the most complex of the methods recommended by ASHRAE. The calculation was done using the in-built method in the Autodesk Revit software where the geometry was designed. Table II is one of the load summaries generated by Autodesk Revit.

#### TABLE II: DEAD REI TRANSFER FUNCTION METHOD

| Variables                          | Values          |
|------------------------------------|-----------------|
| Inputs                             | Values          |
| Area (m²)                          | 455             |
| Volume (m³)                        | 2,121.97        |
| Wall Area (m²)                     | 707             |
| Roof Area (m²)                     | 463             |
| Door Area (m²)                     | 24              |
| Partition Area (m²)                | 0               |
| Window Area (m²)                   | 55              |
| Skylight Area (m²)                 | 0               |
| Lighting Load (W)                  | 6,857           |
| Power Load (W)                     | 4,898           |
| Number of People                   | 500             |
| Sensible Heat Gain / Person (W)    | 67              |
| Latent Heat Gain / Person (W)      | 45              |
| Infiltration Airflow (L/s)         | 68.3            |
| Space Type                         | Classroom/Lecture |
| Output (Calculated Results)        |                 |
| Peak Cooling Load (W)              | 86,905.0        |
| Peak Cooling Month and Hour        | April 2:00 pm   |
| Peak Cooling Sensible Load (W)     | 67,418.0        |
| Peak Cooling Latent Load (W)       | 19,487.0        |
| Peak Cooling Airflow (L/s)         | 5,269.4         |
| Peak Heating Load (W)              | 1,269.0         |
| Peak Heating Airflow (L/s)         | 1,908.5         |

The calculated sensible heat gain is 67,418.00 W, latent heat gain is 19,487.00 W and peak load is 86,905.00 W. These values are close to CLTD/CLF values.

### E. Duct Sizing

Table III shows the calculated sizes of the ducts in the
auditorium which are displayed in the displacement ventilation arrangement in Fig. 3.

### Table III: Ducts Sizes

| Duct Section | Air Quantity (AFR) | AFM capacity (%) | Duct area (%) | Area (m²) | Duct size (mm) |
|--------------|-------------------|------------------|---------------|-----------|----------------|
| Fan to A     | 5,489.81          | 100%             | 100%          | 0.8313    | 1200 x 600     |
| A to 1       | 4,556.54          | 83%              | 87%           | 0.7232    | 1000 x 600     |
| 1 to 2       | 3,052.88          | 67%              | 73.5%         | 0.5316    | 800 x 600      |
| 2 to 3       | 1,526.44          | 50%              | 58%           | 0.3083    | 600 x 400      |
| 3 to 4       | 518.99            | 34%              | 42%           | 0.1295    | 400 x 250      |
| 4 to 5       | 88.23             | 17%              | 24%           | 0.0311    | 250 x 100      |

![Fig. 3. Duct Work](image1)

![Fig. 4. DV Geometry](image2)

![Fig. 5. MV Geometry](image3)

The displacement ventilation geometry was designed with the supply inlets at the sides and the return outlet at the top while the mixed ventilation has its supply inlets on the ceiling and return outlets at the sides.

### G. Mesh

Meshing is necessary for the discretization of the geometry. The meshing breaks up the geometry into nodes and elements at which the calculations are performed. The DV mesh has 206,077 nodes and 1,059,979 elements, while the MV mesh has 171,561 nodes and 867,410 elements.

### H. Governing Equations

The $k - \varepsilon$ model is solved with the governing equations (1) to (4): Equation (1) is the continuity equation, (2) the energy equation, (3) the turbulence kinetic energy equation and (4) is the rate of dissipation equation.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = S_\rho \tag{1}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{v} \varepsilon) = -\nabla \cdot \left( \rho \mathbf{v} \varepsilon \nabla T \right) + \frac{\varepsilon}{\varepsilon} \left[ \left( \frac{\mu + \mu_t}{\sigma_t} \right) \nabla \cdot \left( \frac{k}{\varepsilon} \nabla T \right) + G_1 + G_2 - \rho \varepsilon - Y_M + S_\varepsilon \right] \tag{2}
\]

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{v} k) = \frac{\varepsilon}{\varepsilon} \left[ \left( \frac{\mu + \mu_t}{\sigma_t} \right) \nabla \cdot \left( \frac{k}{\varepsilon} \nabla T \right) + C_{1k} \left( \frac{\varepsilon}{\varepsilon} \right)^{1.5} (G_1 + C_{3k} G_2) - C_{2k}^\prime \frac{k}{\varepsilon} + S_k \right] \tag{3}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{v} \varepsilon) = \frac{1}{\varepsilon} \nabla \cdot \left( \rho \mathbf{v} \varepsilon \nabla T \right) + \frac{\varepsilon}{\varepsilon} \left[ \left( \frac{\mu + \mu_t}{\sigma_t} \right) \nabla \cdot \left( \frac{k}{\varepsilon} \nabla T \right) + C_{1k} \left( \frac{\varepsilon}{\varepsilon} \right)^{1.5} (G_1 + C_{3k} G_2) - C_{2k}^\prime \frac{k}{\varepsilon} + S_\varepsilon \right] \tag{4}
\]

where:
- $G_1$ = generation of turbulence kinetic energy due to the mean velocity gradients;
- $G_2$ = generation of turbulence kinetic energy due to buoyancy;
- $Y_M$ = contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate;
- $\mu_t$ = eddy viscosity;
- $C_{1k}^\prime$, $C_{3k}$, and $C_{2k}^\prime$ are constants; and $\sigma_t$ and $\sigma_\varepsilon$ are turbulent Prandtl numbers for $k$ and $\varepsilon$, respectively; and $S_k$ and $S_\varepsilon$ are user-defined source terms.

$\mu_t$ is evaluated by using Equation (5).

\[
\mu_t = \rho C_{\mu} k^2 \frac{\varepsilon}{\varepsilon} \tag{5}
\]
where \( C_\mu \) is a constant.

I. Boundary Conditions

1) Displacement Ventilation:
Inlet: velocity – 6.604 m/s; gauge pressure – 26.128 Pa; temperature – 292.15 K.
Outlet: gauge pressure – 0 Pa; temperature – 300 K.
Wall: convection; heat transfer coefficient – 1.92 W/m²·K; free stream temperature – 300 K; wall thickness – 0.2 m, material – concrete.
Lecturer: temperature – 309.65 K; material – human-body.
Students: temperature – 309.65 K; material – human-body.
Interior: material – air.

2) Mixed Ventilation:
Inlet: velocity – 6.604 m/s; gauge pressure – 26.128 Pa; temperature – 289.15 K.
Outlet: gauge pressure – 0 Pa; temperature – 300 K.
Wall: convection; heat transfer coefficient – 1.92 W/m²·K; free stream temperature – 300 K; wall thickness – 0.2 m, material – concrete.
Lecturer: temperature – 309.65 K; material – human-body.
Students: temperature – 309.65 K; material – human-body.
Interior: material – air.

J. Visualization
Unity is the visualization software used for a popular game engine with virtual reality capability. Setting up the project requires both the 3D model and simulation results. The 3D model was exported from Autodesk Revit format first into Autodesk 3ds Max then Unity to ensure the textures are readable. The simulation results were not as simple since Unity software does not have an in-built package to read and plot the CSV data format of the result. Hence, the plot functionality had to be programmed.

The algorithm for the code is displayed clearly in Fig. 6.

III. RESULTS AND DISCUSSION
The results obtained from the two ventilation techniques compared are represented in the form of streamlines, volume renderings, temperature and velocity profiles and of course, in a virtual environment. The purpose of the comparison is to determine if the proposed ventilation system (DV) is the better alternative to the existing design (MV). The two systems are evaluated for thermal comfort which includes air purity, air distribution, temperature and humidity.

Figs 7 and 8 show the temperature across the auditorium. The temperature across the room in Fig. 7 is fairly constant at around 293 K which is lower than the temperature (294.5 K) in the room in Figure 8. This implies that DV technique is more efficient than the MV in temperature distribution.

![Fig. 7. DV Temperature Plot](image)

![Fig. 8. MV Temperature Plot](image)

In Fig. 9, the air flows out of the supply diffusers towards the middle of the room and circulates throughout the room. Since the air is supplied at sides of the theatre, the air distribution is focused on the lower portion, which reduces the volume being cooled in DV technique as confirmed by Raman [7]. In Fig. 10, the supply diffusers are fixed on the ceiling while the return diffusers are close to the bottom on either side of the auditorium as also carried out by Abu [22]. The air distribution is not even, there is stagnation of air around the front corner, and there is insufficient air towards the back region. The AC system, in this case, targets the whole volume of the auditorium, which leads to use of more energy in MV technique.
The velocity in the mid-plane of the auditorium (Fig. 11) is between 0.4 – 1.8 m/s. The high-velocity regions are at the supply and return diffusers as expected, with moderate intensity in the mid-front of the auditorium. In Fig. 12, the air delivery is directed downwards, which does not ensure adequate distribution to all areas except each region has its overhead supply diffuser. This implies that the DV technique gives a better velocity profile than the MV.

Fig. 13 shows a portion of the virtual environment with a representation of the simulation results. The result of DV technique is represented in a scattered plot shown in Fig. 13 with three main colours to identify the hot (red), normal (yellow) and cool (green) regions in the auditorium. It should be noted that not all the coordinates were plotted to reduce lag in the system.

IV. CONCLUSION

It can be deduced from the results that the proposed displacement ventilation technique is the better alternative to the mixed ventilation. The displacement ventilation does not require as low supply air temperature as mixed ventilation. Even at this temperature, displacement ventilation can achieve better cooling and this implies energy savings and therefore lower cost. Moreover, the displacement ventilation ensures proper airflow to ensure better air quality than mixed ventilation.

The use of computational fluid dynamics in this numerical analysis has proven to be of considerable help. There was no need to go through the rigour of setting up a laboratory experiment or performing a series of trial and error before obtaining a useful result. The visualization in virtual reality was immersive and convenient compared to the flat computer images.

ACKNOWLEDGMENT

The authors acknowledge the facilities support from Department of Mechanical Engineering, University of Ibadan, Nigeria and the Department of Mechanical Engineering, Obafemi Awolowo University, Ile-Ife.

CONFLICTS OF INTEREST

The authors declare that there is no conflict of interest regarding the publication of this article.

REFERENCES

[1] X. R. Ding, Y. Y. Guo, and Y. Y. Chen, “Design and simulation of an air conditioning project in a hospital based on computational fluid dynamics,” Archives of Civil Engineering, vol. 63, no. 2, pp. 23–38, Sep. 2017.
[2] Y. Hua, G. Huafang, L. Zhenrong, and Z. Bo, “CFD simulation of air-conditioning system in the public area of a metro station and research on energy-saving operation scheme,” Advances in Engineering Research., vol. 135, pp. 699–705, 2017.
[3] A. Scanlon and A. Calderone, “CFD benchmarking: Hamer Hall Auditorium case study,” in Proceedings of Building Simulation: Conference of International Building Performance Simulation Association, pp. 1647–1654, vol. 12, Sydney, 2011.
[4] S. Gilani, H. Montazeri and B. Blocken, “CFD simulation of stratified indoor environment in displacement ventilation: Validation and
sensitivity analysis,” *Building Environment*, vol. 95, pp. 299–313, 2016.

[5] Q. J. Kwong, H. F. Chen, and A. A. Razak, “Computational simulation of indoor thermal environment in a tropical educational hall with displacement ventilation,” *Peranakan Journal of Science and Technology*, vol. 25, pp. 77–88, Jan. 2017.

[6] W. K. Chow and T. Hong, “Application of computational fluid dynamics in building services engineering,” *Appl. Comput. Fluid Dyn. Build. Serv. Eng.*, vol. 31, no. 5, pp. 425–436, 1996.

[7] M. M. E. Ramlan, “Computational fluid dynamics analysis of HVAC system in auditorium,” *International Journal of Advanced Research and Development*, vol. 1, no. 5, pp. 68–72, May 2016.

[8] L.N., Cao, J. Cao, S. Lee, Y. Zhang, and X. Tie, “Numerical simulation of the micro environment in the Han Yang Mausoleum Museum,” *Aerosol Air Qual. Res.*, vol. 12, pp. 544-552, July 2012.

[9] M. M. Tap, H. M. Kamar, A. K. Marsono, N. Kamsah, and K. A. M. Salim, “Simulation of thermal comfort of a residential house,” *International Journal of Computer Science Issues*, vol. 8, no. 5; pp. 200-208, Sep. 2011.

[10] Q. Li, H. Yoshino, A. Mochida, B. Lei, Q. Meng, L. Zhao, and Y. Lun, “CFD study of the thermal environment in an air-conditioned train station building,” *Building and Environment*, vol. 44, no. 7, pp. 1452–1465, 2009.

[11] Y. Won, M. S. Waring, and D. Rim, (2019). “Understanding the spatial heterogeneity of indoor OH and HO2 due to photolysis of HONO using Computational Fluid Dynamics (CFD) simulation,” *Environmental Science & Technology*, vol. 53, pp. 14470–14478, Nov. 2019.

[12] M. J. Banjac, “Application of computational fluid dynamics in cooling systems design for special purpose objects,” *FME Transactions*, vol. 42, no. 1, 26-33, Jan. 2014.

[13] M. Hosokawa, T. Fukuda, T. Michikawa and A. L. I. Motamedi, “Integrating CFD and VR for indoor thermal environment design feedback,” *The 21st International Conference on Computer-Aided Architectural Design Research in Asia (CAADRIA) 2016*, pp. 663–672, Melbourne, 2016.

[14] C. S. Sanchez, and Z. Xiaojin, “Visualization of CFD simulation results in VR environment for design feedback.” Presented at Building Performance Analysis Conference and SimBuild co-organized by ASHRAE and IBPSA-USA Chicago, IL, USA, September 26-28, 2018.

[15] J.-R. Lin, J. Cao, J.-P. Zhang, C. van Treeck, and J. Frisch, “Visualization of indoor thermal environment on mobile devices based on augmented reality and computational fluid dynamics,” *Automation in Construction*, vol. 103, pp. 26–40, 2019.

[16] M. Kim, S. Yi, D. Jung, S. Park, D. Seo, “Augmented-reality visualization of aerodynamics simulation in sustainable cloud computing,” *Sustainability*, vol. 10, 1-14, April 2018.

[17] T. Fukuda, K. Yokoi, N. Yabuki, and A. Motamedi, “An indoor thermal environment design system for renovation using augmented reality,” *Journal of Computational Design and Engineering*, vol. 6, pp. 179–188, 2019.

[18] J. M. D. Delgado, L. Oyedele, P. Demian, and T. Beach, “A research agenda for augmented and virtual reality in architecture, engineering and construction,” *Advanced Engineering Informatics*, vol. 45, pp. 1-21, June 2020.

[19] M. Berger, and V. Cristie, “CFD post-processing in Unity3D,” *Procedia Computer Science*, vol. 51, pp. 2913–2922, June 2015.

[20] J. Moreland, J. Wang, Y. Liu, F. Li, L. Shen, B. Wu, and C. Zhou, “Integration of augmented reality with computational fluid dynamics for power plant training.” Presented at the International Conference on Modeling, Simulation and Visualization Methods, Las Vegas, NE, USA, July 22–25, 2013.

[21] R. A. Parsons, *ASHRAE Handbook: Fundamentals, SI edition*, Atlanta, GA, USA: America Society of Heating, Refrigeration and Air-Conditioning Engineering Inc., 1997, Chapter 27 and 30.

[22] R. Abu, “Ducting analysis of central air-conditioning system of Faculty of Technology Lecture Theatre,” B.Sc. project report, Dept. of Mech. Eng., Univ. of Badan, Badan, Badan, Nigeria, 2001.